FINAL REPORT

GEOROC: A NUMERICAL MODELLING PACKAGE FOR DESIGNING UNDERGROUND OPENINGS IN POTASH

by

A. F. Fossum, R. R. Brander,
J. D. Chieslar and H. Y. Tammemagi.
RE/SPEC Ltd.
Calgary, Alberta.

MAY 13 1985
MICROMEDIA

February, 1985

132 pp

Work on this project was conducted under the auspices of the Canada Centre for Mineral and Energy Technology, Energy, Mines and Resources, Canada.

DSS File No.:	26SQ.23440-3-9155
DSS Contract Serial No.:	OSQ83-00269
CANMET Project No.:	310104
CANMET Scientific Authority:	Y. S. Yu
RE/SPEC Ltd. Report No:	RSL-006

TABLE OF CONTENTS

		Page
	EXECUTIVE SUMMARY (English)	
	EXECUTIVE SUMMARY (Francais)	
1	INTRODUCTION	1
2	SALT/POTASH PROPERTIES	2
	2.1 CONSTITUTIVE CREEP LAWS	2
	2.2 FAILURE CRITERIA FOR SALT/POTASH	10
3	GEOROC	13
	3.1 INTRODUCTION	13
	3.2 CAPABILITIES AND LIMITATIONS	13
	3.3 MATHEMATICAL BASIS	16
	3.4 COMPUTER IMPLEMENTATION	18
	3.5 COORDINATES AND UNITS	22
	3.6 OUTPUT	22
4	PRE- AND POST-PROCESSORS	25
	4.1 GEOMESH	25
	4.2 GNP	25
	4.3 NCAR	26
5	USER'S GUIDE	28
	5.1 GEOROC	28
	5.2 GNP USER GUIDE	42
	5.2.1 ALGEBRA	42
	5.2.2 TPLOT	46
	5.2.3 SPLOT	47

TABLE OF CONTENTS
(Continued)

		Page
6	VERIFICATION AND EXAMPLE PROBLEMS	52
	6.1 CANTILEVER BEAM	52
	6.2 THICK WALLED CYLINDER	52
	6.3 BENCH MARK ANALYSIS	56
	6.4 POTASH MINE SIMULATIONS	63
7	CONCLUSIONS/RECOMMENDATIONS	83
8	REFERENCES	85
9	ACKNOWLEDGMENTS	87
	APPENDIX A AN ALPHABETICAL LIST BRIEFLY DESCRIBING ALL GEOROC SUBROUTINES AND FUNCTIONS	
	APPENDIX B USER'S MANUAL FOR PROGRAM GEOMESH	

EXECUTIVE SUMMARY

The development of a computer program which can accurately model the behaviour of underground potash mines would have many benefits for the potash industry. It would provide a tool for quantitative assessment of the stability and, hence, safety of proposed new layouts, particularly novel concepts such as multi-seam mining or longwall extraction, which would otherwise be difficult to assess. However, the analysis of stress and displacement in a complex, time-dependent (viscoelastic) material such as salt and potash is very difficult and it has only been in the past several years that computer technology has developed to the point where such problems are solvable at a reasonable cost.

CANMET awarded RE/SPEC Ltd. a contract to develop a numerical modeling software package for use in potash mine design and with the capability of modeling viscoelastic material and identifying room convergence, convergence rates, ground stresses and contain a suitable failure criterion.

GEOROC is the resultant computer program. It is a large and sopnisticated finite element stress analysis program which can analyze a wide range of potash and other underground mining problems. Single or multiple openings of complex shape and varied excavation sequence can be modeled. Elastic and/or viscoelastic materials with complex geologic stratification can be simulated.

GEOROC has the following capabilities:

1. Geometry: Two dimensional (plane strain and plane stress) and axisymmetric.

2. Materials: Isotropic elastic and/or viscoelastic material properties. Each element in the mesh may have it's material specified. Three constitutive laws describing creep material behaviour are incorporated in GEOROC. The power law is an

empirical law which is most suitable for describing the initial, transient portion of creep. The Norton law is suitable for modeling steady state creep and has received relatively widespread use in modeling the behaviour of salt mines and solution mined caverns in salt. The exponential-time law is also incorporated and models both transient and steady state creep.

3. Mining Sequences: Initial and sequential excavations and/or backfill can be simulated.

4. Pressure Loading: Pressures may be applied to the interior walls of the excavations for simulation of support system, storage of fluids or backfill.

5. Boundary Conditions: Kinematic (displacement) constraints, external surface loading, skewed boundary orientation relative to global axes are incorporated.

6. Initial Stress State: Based on overburden specific weight and lateral earth pressure coefficient, or based on body forces.

7. Factor of Safety: Two different formulations of factor of safety are incorporated. The first is the Mohr Coulomb criterion and is suitable for assessing brittle failure in elastic rocks. The second is based on total creep strain and is suited for assessing potential failure in viscoelastic materials.

8. Mechanical properties of selected elements may be altered during the analysis. This feature is intended for modeling backfill behaviour.

GEOROC has been developed on a VAX 11/750 using the VMS operating system and has been written in FORTRAN 77 so that it can be installed on different computers using different operating systems with a minimum of effort.

The finite element method is used by GEOROC with eight-noded elements. Solution of problems involving time-dependent deformation is accomplished using an incremental procedure which allows nonlinear creep problems to be solved as a successive series of linear problems for each time step.

The preprocessor program, GEOMESH, has been written to assist the user in preparing the mesh data. GEOMESH, which is based on the concept of mapping using isoparametric shape functions, is quite flexible and allows a variety of complex meshes to be generated with relative ease.

A package of three postprocessor programs (ALGEBRA, TPLOT and SPLOT) collectively known as GNP (for Graphs and Plots) is used to help the user inspect the large mass of output data which is produced by GEOROC. TPLOT generates x versus time plots, where x is an appropriate output variable, such as stress or displacement, selected by the user. SPLOT generates x versus distance plots. ALGEBRA allows the user to create functions of GEOROC's output data (for example, calculate a factor of safety) which in turn can be plotted by TPLOT and SPLOT.

Since GEOROC is a complex program it is essential that it be thoroughly tested. To this end, GEOROC was used to model the behaviour of two models: The first problem was an elastic, cantilever beam which is fixed at one end and has a load at the other end. The second verification problem consisted of a thick-walled, infinitely long, hollow cylinder with an applied internal pressure loading. The cylinder consisted of a viscoelastic material governed by a power law. For both these problems, the stresses and displacements predicted by GEOROC are in excellent agreement with the analytical solutions.

To illustrate how GEOROC may be used for analyzing rock mechanics problems which are relevant to the potash industry, a hypothetical room/pillar layout based on Cominco mine was analyzed. The model consists of a five-room panel and incorporates the Dawson Bay Formation as well as the evaporites, and the rooms are sequentially excavated. Two cases were investigated. In the first, the rooms were excavated and left empty, while in the second potash tailings were used as backfill.

GEOROC is a powerful and sophisticated stress analysis tool which has application not only in the potash industry but in a broad range of underground mining and engineering situations. It is recommended that continued evolution of GEOROC be implemented to increase its capabilities and improve its computational efficiency. It is also recommended that a detailed study of a Saskatchewan potash mine be undertaken in which GEOROC analyses are integrated with laboratory derived material properties and with underground closure and stress data.

COMPTE-RENDU DE LA DIRECTION

La mise au point d'un programme d'ordinateur offrant un modèle précis du comportement des mines souterraines de potasse serait d'un grand bénéfice pour l'industrie de la potasse. Grâce à cet outil, une évaluation quantitative de la stabilité et du même coup, de la sécurité de nouveaux aménagements proposés devient possible et plus particulièrement, l'évaluation de nouvelles idées, telles l'exploitation par couches multiples ou l'exploitation par longue taille qu'il serait difficile d'évaluer autrement. Cependant, l'analyse des forces de tension et du déplacement dans des matériaux complexes se modifiant avec le temps (viscoélastiques) comme le sel et la potasse est très difficile et ce n'est que depuis quelques années que la technologie des ordinateurs a progressé jusqu'au point où de tels problèmes sont solubles à un coût raisonnable.

CANMET a accordé à RE/SPEC Ltd. un contrat pour la mise au point d'un logiciel de modélisation numérique servant à l'élaboration de plans d'aménagement de mines de potasse, pouvant procéder à la modélisation d'un matériau viscoélastique et à l'identification de l'affaissement de la chambre, des taux d'affaissement ainsi que des tensions du sol et contenant un critère adéquat de rupture.

GEOROC est le programme d'ordinateur résultant. Ce programme d'analyse des forces de tension par la méthode des éléments finis est imposant et hautement perfectionné; il peut analyser une grande variété de problèmes rencontrés lors de l'exploitation des mines de potasse et autres mines souterraines. Des modèles d'ouvertures simples ou multiples de forme complexe et de séquence variée d'excavation peuvent être créés. Des matériaux élastiques aussi bien que viscoélastiques possédant des stratifications géologiques complexes peuvent être simulés.

GEOROC possède les capacités suivantes:

1. Géométrie: À deux dimensions (plan de contrainte et plan de force) et axisymétrique.

2. Matériaux: Propriétés de matériaux isotropiques élastiques aussi bien que viscoélastiques. Les matériaux de chaque élément de la maille peuvent être spécifiés. Trois lois constitutives décrivant le comportement du fluage d'un matériau sont inclus dans GEOROC. La loi de puissance est une loi empirique qui est surtout adéquate pour représenter la partie initiale et transitoire du fluage. La loi de Norton est appropriée pour la modélisation du fluage à régime permanent et son usage est relativement répandu dans la modélisation du comportement des mines de sel et des cavernes de sel exploitées par dissolution. La loi de temps-exponentiel a été aussi incluse et sert à la modélisation du fluage à régime transitoire aussi bien qu'à celle du fluage à régime permanent.

3. Séquences d'exploitation minière: Des excavations initiales, et successives et/ou le remblayage peuvent être simulés.

4. Charges de pression: Des pressions peuvent être appliquées aux murs intérieurs des excavations pour simuler la présence de systèmes de soutènement, d'un stockage de fluides ou d'un remblai.

5. Conditions aux limites: Les contraintes cinématiques (déplacement), la charge maximale de la surface externe, l'orientation inclinée limite relative à l'ensemble des axes, ont été incluses.

6. État initial de tension: Basé sur le poids spécifique de la couverture et le coefficient de poussée latérale des terres ou bien basé sur les forces des corps en présence.

7. Coefficient de sécurité: Deux formulations différentes du coefficient de sécurité sont incluses. La première est le critère de Mohr-Coulomb et est convenue pour l'évaluation de la rupture fragile dans les roches élastiques. La deuxième est basée sur la déformation totale du fluage et est convenue pour évaluer une

rupture possible dans les matériaux viscoélastiques.

8. Les propriétés mécaniques d'éléments choisis peuvent être changées au cours de l'analyse. Cette caractéristique a pour but la modélisation du comportement du remblayage.

GEOROC a été mis au point sur un VAX 11/750 en utilisant le système d'exploitation VMS et est écrit en FORTRAN 77 de sorte que son installation sur différents ordinateurs utilisant différents systèmes d'exploitation n'exige que peu d'effort.

GEOROC utilise la méthode des éléments finis avec huit éléments liés. Les problèmes incluant une déformation en fonction du temps sont résolus grâce à une procédure incrémentielle qui permet de résoudre les problèmes de fluage non linéaire comme une succession de problèmes linéaires pour chaque subdivision de temps.

Le préprocesseur, GEOMESH, a été écrit pour venir en aide à l'utilisateur lors de la préparation des données de la maille. GEOMESH, qui est basé sur le concept de correspondance utilisant des fonctions de formes isoparamétriques, est assez flexible et permet de créer aisément une variété de mailles complexes.

Un bloc de trois postprocesseurs (ALGEBRA, TPLOT et SPLOT), mieux connus sous le nom de GNP (pour "Graphs and Plots"), sert à assister l'utilisateur lors de l'inspection de la grande quantité de données de sortie produites par GEOROC. TPLOT trace les courbes de x en fonction du temps dans lesquelles x est une variable de sortie appropriée que choisit l'utilisateur, comme par exemple la force de tension ou le déplacement. SPLOT trace les courbes de x en fonction de la distance. ALGEBRA permet à l'utilisateur de créer des fonctions à partir des données de sortie de GEOROC (par exemple, le calcul d'un coefficient de sécurité), lesquelles peuvent à leur tour être tracées par TPLOT et SPLOT.

Puisque GEOROC est un programme complexe, il devient essentiel de le tester à fond. À cette fin, GEOROC a été utilisé pour la modélisation du comportement de deux modèles des éléments finis: Le premier problème portait sur une poutre élastique en porte-à-faux, dont l'une des extrémités

est fixe et l'autre porte une charge. Le deuxième problème de vérification consistait en un cylindre creux à paroi épaisse, d'une longueur infinie, auquel une pression interne de charge est appliquée. Les matériaux composant le cylindre étaient viscoélastiques et obéissaient à la loi de puissance. Pour ces deux problèmes, les prédictions de GEOROC quant aux forces de tension et aux déplacements concordent de façon excellente avec les solutions analytiques.

Pour donner un exemple de la façon dont GEOROC peut être utilisé pour l'analyse de problèmes de mécanique des roches qui sont pertinents à l'industrie de la potasse, un agencement hypothétique de chambres et piliers basé sur la mine Cominco a été analysé. Le modèle consiste en un groupe de cinq chambres isolées, excavées en succession, et comprend la Formation de la Baie de Dawson aussi bien que les évaporites. Deux cas furent examinés. Dans le premier, les chambres furent excavées et laissées vides, tandis que dans le deuxième, des résidus d'exploitation de potasse furent utilisés comme remblai.

GEOROC est un outil performant et complexe d'analyse des forces de tension qui a des applications non seulement dans l'industrie de la potasse mais aussi dans une grande variété de situations propres aux mines souterraines et à l'ingénierie. Il est suggéré qu'un développement continu de GEOROC soit assuré pour augmenter ses capacités et améliorer l'efficacité de ses calculs. Il est également suggéré qu'une mine de potasse en Saskatchewan fasse l'objet d'une étude détaillée qui amalgamerait les analyses de GEOROC, les résultats en laboratoire sur les propriétés des matériaux et les données sur la constriction et les forces de tension souterraines.

1 INTRODUCTION

The layouts in the potash mines of Saskatchewan have evolved since the early 1960's largely by a process of trial and error. Although this evolutionary approach to mine design is relatively safe, it does not allow new concepts such as total extraction or multi-level mining to be readily assessed. Nor does it provide a fundamental understanding of the rock mechanics principles which govern the underground mine behaviour. CANMET recognized that the development of a computer program which could suitably model excavations in salt and potash would greatly benefit the potash industry. However, the analysis of stress and displacement in a complex time dependent (visco-elastic) material such as potash/salt is very difficult and it has only been in the past several years that computer technology has developed to the point where such problems are solvable at a reasonable cost.

In January, 1984 CANMET awarded RE/SPEC Ltd. a contract to develop a numerical modeling software package to design underground openings in potash. The software package needed to be capable of modeling visco-elastic material and identifying room convergence, convergence rates, ground stresses and contain a suitable failure criterion.

GEOROC is the resultant computer program. It is a large and sophisticated finite element stress analysis program which can analyze a wide range of potash and other underground mining problems. Single or multiple openings of complex shape and varied excavation sequence can be modeled. Elastic and/or visco-elastic materials may be included and complex geologic stratification may be specified. RE/SPEC has also written a separate mesh generator program (GEOMESH) to assist with the preparation of the input data. In addition, two software packages (GNP and NCAR) are provided which allow the user to review, in graphical form, selected portions of the large volume of output data which GEOROC produces.

The purpose of this report is to describe GEOROC and its supporting programs. Section 2 reviews the failure criteria and constitutive laws which are used to describe the material behaviour of salt/potash. Section 3 describes the capabilities and limitations of GEOROC and also

provides an overview of its mathematical formulation and computer implementation. The supporting programs (GEOMESH, GNP and NCAR), also known as pre- and post-processors, are described in Section 4. Detailed instructions on how to prepare the input file for GEOROC and run the program are presented in Section 5. This section is intended to be self-contained and can be used independently as a user's guide. Section 6 contains two verification test examples and one hypothetical mining problem which have been analyzed by GEOROC. The first is a simple, elastic beam; the second is a thick-walled visco-elastic cylinder subjected to internal pressure. Both these problems have closed-form solutions for comparison. The final problem is typical of potash mining and consists of a five room panel which has been modeled on Cominco Mine. Two analyses have been performed, with and without crushed salt backfill. These problems form an important part of this report since they provide not only verification that GEOROC is functioning correctly, but also illustrate the scope and range of problems which may be asessed, as well as demonstrating how to prepare input for these problems. The final section briefly describes future work which might be undertaken to improve the capability to numerically model underground openings in potash.

2 SALT/POTASH PROPERTIES

2.1 CONSTITUTIVE CREEP LAWS

Although salt has a relatively simple ionic structure, its mechanical behaviour is very complex. Under loading rock salt exhibits a viscoelastic response, that is, it creeps with time. This behaviour is non-linear and, thus, introduces difficulties into the mathematical formulations for stress-strain.

The deformation of rock salt is a function of the stress to which it is subjected. The hydrostatic portion of the stress results in elastic deformation and can be readily expressed in terms of the elastic moduli. Typically, Young's modulus is about 30 GPa and Poisson's ratio is about 0.35, and these values do not vary considerably from salt to salt (Pfeifle et al., 1982 and Hardy, 1982). The non-hydrostatic (or deviatoric) part of

the stress causes strains that are not recoverable, that is, creep with time. The remainder of this section discusses the constitutive laws which are used to describe this creep behaviour. The discussion is for salt since sylvite and other potash minerals are expected to behave in the same general manner, although the latter have received very little study. Furthermore, the volumes of salt at the mining horizons in Saskatchewan are much greater than the volumes of potash. Other viscoelastic materials which have received considerable research attention and which may provide useful insights include ice (e.g., Sego and Morgenstern, 1983) and metals at high temperatures (e.g., Webster et al., 1969).

Stresses induced by mining cause creep of salt because it is a relatively soft rock. The deformation is caused, at the atomic scale, by the motion of defects such as dislocations, point defects and individual atoms within the cystals. The formation and migration of these defects are hindered by potential barriers and can be analyzed by statistical mechanics and thermodynamic principles. However, the derivation of constitutive laws directly from such basic principles has not been achieved to date, due to the complexity of the problem, and virtually all laws are empirically-based.

Figure 2-1 shows the axial strain of salt as a function of time under constant stress, and it is seen that creep occurs in three stages. Primary (or transient) creep exhibits a high rate of deformation which decreases with time. The rate is then constant (secondary or steady state creep) for some period of time. After sufficient time and with appropriate stress and temperature conditions, the rate of deformation increases rapidly and the salt undergoes creep rupture (tertiary creep). Tertiary creep seldom occurs in laboratory experiments and generally does not contribute significantly to salt deformation under most mining and geological situations. For these reasons, tertiary creep is not included in the constitutive laws which are discussed below.

Comprehensive reviews of constitutive laws for salt creep have been presented by Senseny (1981) and by Carter (1981). A brief summary of some of the laws is given here and then a particular law is recommended for use in GEOROC for use in analysis of the potash mines in Saskatchewan.

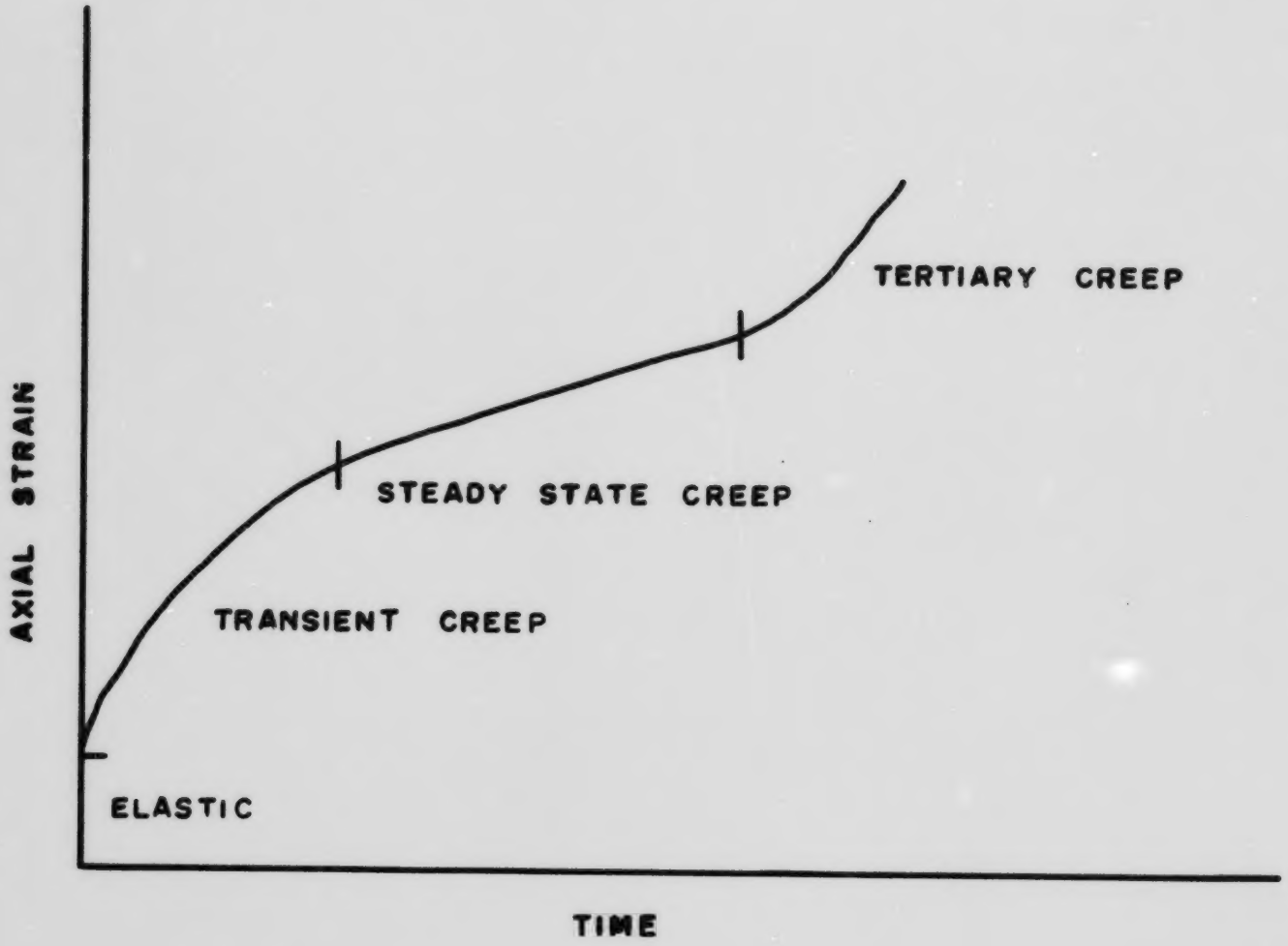


Figure 2-1. Schematic Representation of Creep Strain as a Function of Time.

Clearly, the movement of defects and, hence, creep of potash/ salt is dependent on temperature. This dependence is generally either in exponential form, $e^{-Q/RT}$ where Q is the free energy of activation, R is the universal gas constant and T is absolute temperature, or is expressed as a power law, that is, T^m. However, temperature can be assumed to be constant for the potash mining applications which are of relevance to this study and, hence, both forms reduce to a constant. For this reason, the temperature dependence of creep will be omitted in the ensuing discussion.

The power law is an empirical formula commonly used to describe transient creep. The law is expressed as:

$$e = As^m t^n \qquad (2.1)$$

where e is a scalar measure of transient creep strain, s is axial stress difference, t is time since loading and A, m, and n are experimentally determined parameters. Note that the temperature dependence has been incorporated into the parameter A, since we are dealing with an isothermal situation. The above law holds only for constant stress and more complex stress histories are handled using either a time hardening or a strain hardening strategy. Figure 2-2 illustrates these two strategies. Creep occurs at s_1 until t* when it is increased to s_2. The subsequent creep strain is described by the 'time hardening' curve which is obtained by lowering the s_2 curve to the s_1 curve at $t = t^*$ or the 'strain hardening' curve which is obtained by shifting the s_2 curve to the right until it contacts the s_1 curve at $t = t^*$.

The exponential-time law models both transient and steady-state creep and is expressed as (Senseny, 1981):

$$\begin{aligned} e &= \dot{e}_{ss}t + ea[1 - \exp(-B\dot{e}_{ss}t)] && \text{for } \dot{e}_{ss} \gg \dot{e}^* \\ e &= \dot{e}_{ss}t + ea.\dot{e}_{ss}/\dot{e}^*[1 - \exp(-B\dot{e}^*t)] && \text{for } \dot{e}_{ss} < \dot{e}^* \end{aligned} \qquad (2.2)$$

where e = creep strain
s = deviatoric stress
t = time

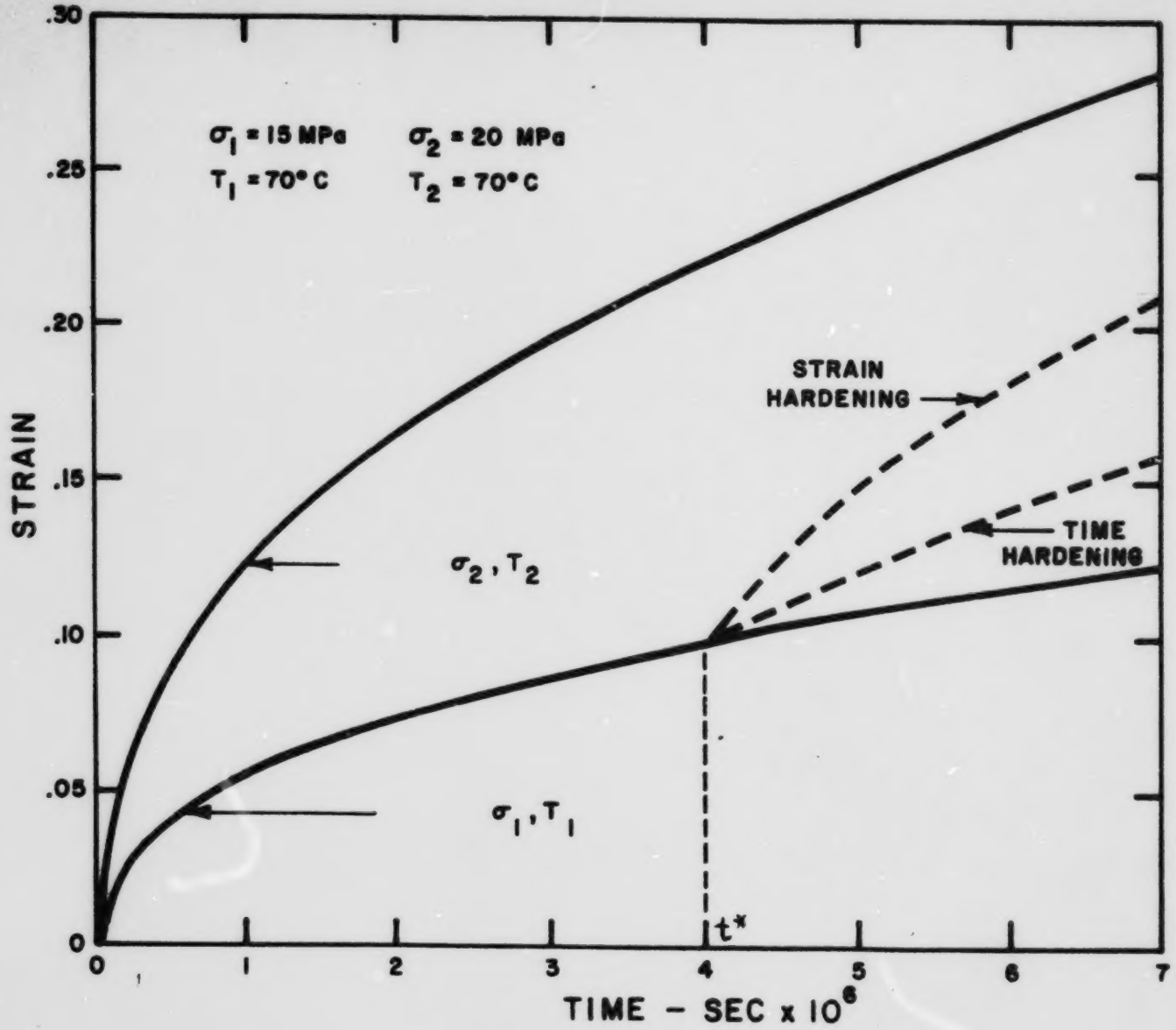


Figure 2-2. Comparison of Strain-Hardening Strategies to Account for Creep Strain under Non-Constant Stress and Temperature Histories.

$\dot{e}_{ss} = As^n$ (steady state strain rate)
e_a, B, n = parameters to be determined experimentally.
$\dot{e}^*$ = critical strain rate.

History, or stress changes, can be accomodated by using a strain hardening procedure. Laboratory testing has shown that the choice of the two relationships in Equation (2.2) depends on whether the steady-state strain rate is greater or less than the critical strain rate.

A creep law introduced by Gangi (1981) assumes that a number of mechanisms contribute independently to the deformation. His multiple exponential-time creep law is given as:

$$e = \dot{e}_{ss}t + \sum_{i=1} b_i(1-\exp(-c_i t) \qquad (2.3)$$

where e is the total strain, b_i is the transient creep strain associated with the i^{th} mechanism, e_{ss} is the steady-state creep rate, m is the number of creep mechanisms, and t is the time since the application of stress. At this time, more study is required to determine if the multiple exponential-time model has any significant merits relative to the exponential-time law.

For many practical applications, a steady-state constitutive law may be adequate. For constant temperature, the Norton power law is expressed as)

$$\dot{e}_{ss} = As^m \qquad (2.4)$$

where s is the axial stress difference, e_{ss} is the steady state creep rate and A and m are parameters to be determined experimentally. M is usually in the range of 3 to 6 (Carter, 1981). The Norton power law has been used to model the behaviour of salt mines (Van Sambeek, 1984) and solution mined caverns in salt (Hardy, 1982).

Other, more complex, constitutive laws have been posed such as the deformation mechanism law (Munson and Dawson, 1979) and the unified creep-plasticity law (Krieg, 1980) but will not be discussed here.

To provide the reader with an idea of typical values for parameters in the various creep laws, Table 2-1 lists experimentally determined parameter values for the Norton power law, the power law and the exponential-time law for typical salts.

All of the preceding constitutive laws have some deficiencies. First, although all the laws have methods for handling stress changes, none of them model complex histories adequately. For example, for stress increasing laboratory experiments, the time hardening formulation underestimates the measured strain whereas the strain hardening formulation overestimates it. For stress-decreasing laboratory experiments, the predictive capability of these laws is not as good as it is for stress-increasing experiments.

Another deficiency is that all the laws discussed so far are one-dimensional (or scalar) formulations. For proper structural analysis, three-dimensional formulations are necessary using strain and stress tensors. To extend the one-dimensional laws to three-dimensional forms, a scalar creep potential must exist which relates the deviatoric stress and strain. Although the existence is assumed, it has not been rigorously proven

The third deficiency is that the laws are empirical and are based largely on laboratory data. Thus, their extrapolation to in-situ mining conditions must be treated with some caution. For example, studies by Hardy (1982) tentatively indicate that there is a size effect associated with creep of salt in that the creep rate tends to decrease as sample size increases.

It is recommended that GEOROC contain the capability to model both transient and steady state creep. It is recommended that both the power law of Equation 2.1 and the Norton power law, Equation 2.4, be incorporated and that the user may specify either one or both for his analysis. These laws have been selected because:

1) they are simple and thus will require relatively few parameters to be defined. This should also result in more efficient computing;

2) both laws have received widespread use and, thus, there is a

Table 2-1. Examples of Creep Law Parameter Values for Various Salts*

1. Norton Power Law (Equation 2.4)

$A = 1.7 \times 10^{-16}$ $[MPa^{-n}\ s^{-1}]$
$m = 5.7$

For Lotsberg Formation salt, Alberta (unpublished RE/SPEC data)

2. Power Law (Equation 2.1)

$A = 3.78 \times 10^{-8}$ $[MPa^{-m}\ s^{-n}]$
$m = 2.64$
$n = 0.38$

For Avery Island salt dome, Louisiana (Mellegard et al., 1981)

3. Exponential Time Law (Equation 2.2)

$A = 8.49 \times 10^{-15}$ $[MPa^{-n}\ s^{-1}]$
$n = 4.15$
$B = 180$
$e_a = 5.50 \times 10^{-2}$

$\dot{e}^* = 5 \times 10^{-8}$ $[s^{-1}]$

For Permian Basin, cycle 5 salt (Pfeifle et al., 1982), at 300 K, only valid for $\dot{e} > 5 \times 10^{-8}\ s^{-1}$.

* If No Units are Shown, The Parameter is Dimensionless.

relatively large data-base of case histories and laboratory experiments from which to draw (for example, Hardy, 1982).

3) they can accommodate stress changes (both increases and decreases).

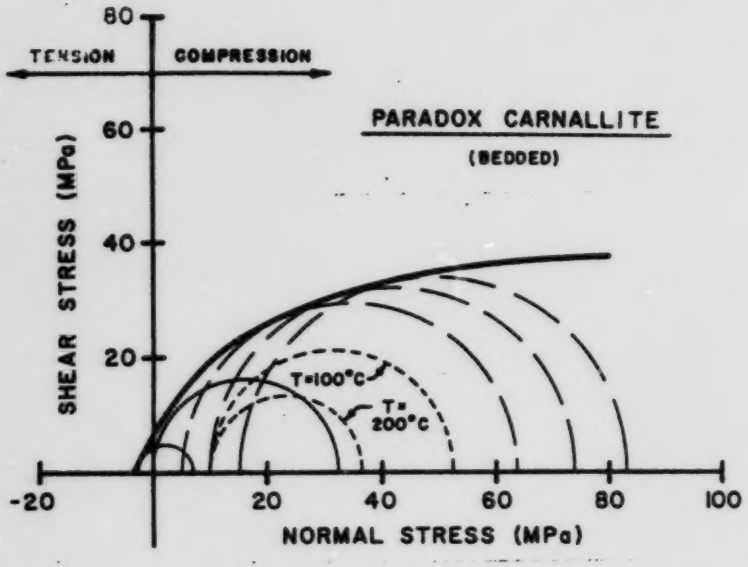
As a final precaution, it must be stressed that the choice of constitutive law is only one step in the prediction of mine deformations. Laboratory studies by Pfeifle et al., (1982) and Hardy (1982) have shown that the creep of salt from different geographic locations can vary significantly. This underlines the importance of carefully determining the constants to be used in the laws.

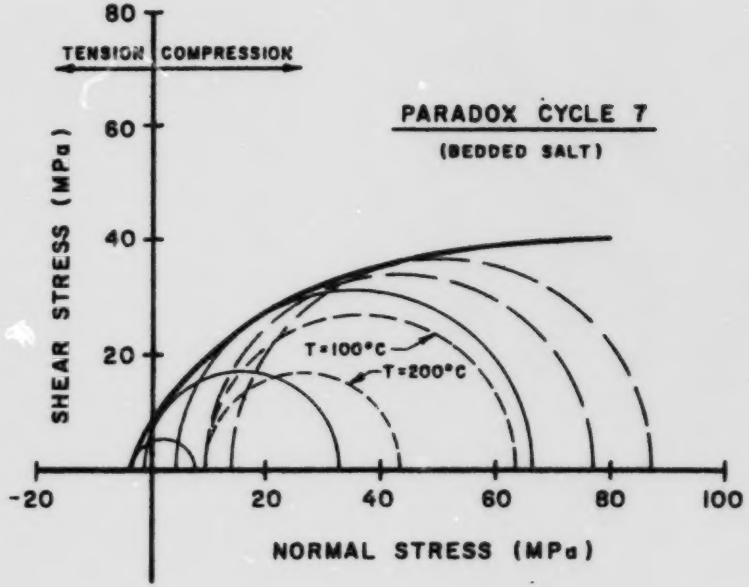
2.2 FAILURE CRITERIA FOR SALT/POTASH

The failure of salt, be it by fracturing, crushing or accelerated (tertiary) creep, has received relatively little attention. The mechanisms of failure are complex and time-dependent with the salt generally yielding (i.e. creeping) rather than failing under stress. No satisfactory failure criteria for salt are presently available. A review of this topic is presented by Carter (1981).

Failure of salt under quasi-static testing conditions has been studied by a number of groups (Wawersik and Hannum, 1979, 1980; Hansen and Mellegard, 1979; Pfeifle et al, 1982). In general, the results indicate that unconfined specimens deformed at room temperature fail at about 2 - 5% axial strain and about 20 - 25 MPa pressre. An increase in confining stress results in a pronounced increase in strength and total strain at failure. An increase in temperature results in a decrease in strength and an increase in total strain at failure. Mohr's circles for the state of stress at failure are shown in Figure 2-3 for salts from the Paradox Basin, Utah and Permia Basin, Texas. It is seen that the failure envelope is <u>not</u> linear. The results in this paragraph are for quasi-static testing and, thus, effectively only apply to time-independent deformation.

Wawersik and Hannum (1979) observed accelerating (tertiary) creep in several of their constant stress (creep) experiments. They noted that accelerating creep was accompanied by increases in volumetric strain (dilatancy). Based on their observations, Wawersik and Hannum (1980) have





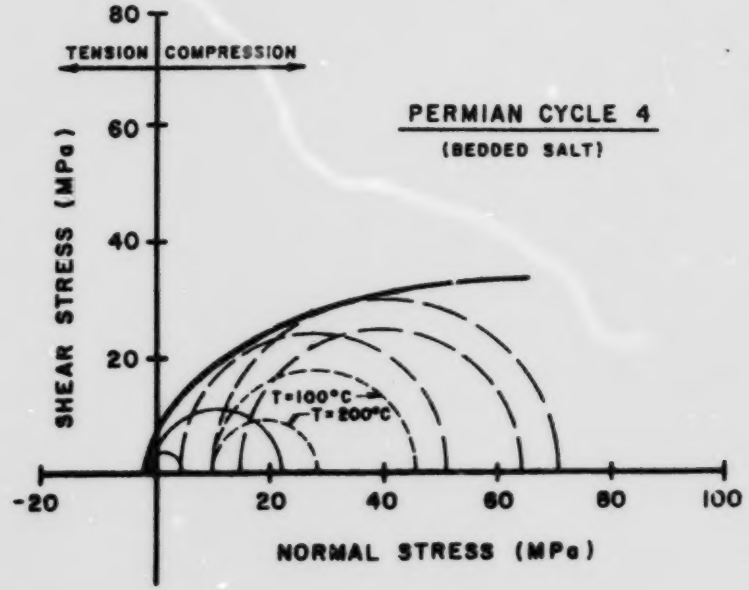


Figure 2-3. Mohr's Circles at Failure for Salt and Carnallite.

suggested that there are limiting creep strains that can be sustained prior to failure at given physical conditions and that these may be determined from quasi-static mechanical behaviour. That is, under similar conditions a salt specimen deformed under quasi-static conditions. The basis for this empirical hypothesis is that there is some limiting damage to specimen integrity before fracture instability ensues which is related to the amount of strain incurred.

Preece and Wawersik (1984) have proposed a failure criterion which is based solely on creep strain and which has been used in nuclear waste repository studies in bedded salt. The failure of salt is described by a failure function, F, such that when F becomes positive, the salt fails. The function is considered to be conservative and is given by

$$F = e - 0.023 - f(p) \qquad (2\text{-}5)$$

where

$$f(p) = \begin{cases} 0.132 & p \geq 6 \text{ MPa} \\ 0.0443p - 0.0037\,p^2 & p < 6 \text{ MPa} \end{cases}$$

e is creep strain, and p is the negative of the trace of the stress tensor (the pressure) in MPa, and is positive in compression. The function F can be used for time dependent (creep) analysis since it depends only on creep strain. Although the parameters in Equation 2-5 were derived specifically for salt from the Permian Basin in south eastern New Mexico, the expression is probably valid for most other salts as well, since it is conservative.

Two failure criteria are implemented in GEOROC. The first is a linear Mohr-Coulomb failure criterion which can be used to model quasi-static failure. This criterion can be applied when doing the post-processing of results using the GNP software (see Section 4 for details). This criterion may also be applied to other (non-salt) rock which behave elastically. Due to the uncertainty in the understanding of salt failure mechanics it was felt that more complex quasi-static failure criteria were not warranted, although such can readily be programmed into either GEOROC or GNP, should they be required for special applications. The second failure criterion is that described in Equation 2-5 and it is implemented directly in GEOROC (see Section 5). It should be noted that there are no provisions for modeling the post fracture behaviour and calculating or redistributing stress. The failure criteria simply indicate when fracturing of the rock salt might be possible.

3 GEOROC

3.1 INTRODUCTION

GEOROC has been developed specifically for analysis of structural problems associated with underground salt mining. The program analyzes mine geometries and predicts stress displacement and failure distribution in space and time. Salt and potash are rate-dependent materials that deform with time when subjected to deviatoric stress as described in Section 2.1. Three constitutive laws describing creep material behaviour (the power, Norton and time-exponential laws) are incorporated in GEOROC and changing stress conditions are handled in the power law using the time-hardening strategy described in Section 2.1. The time-exponential law uses a history function which is effectively a strain-hardening procedure. In addition GEOROC has the capability to model elastic behaviour. The program is quite flexible and can also be used to analyze mines or excavations in elastic (non-creeping) rock types. In addition, GEOROC can be used to study a number of engineering problems such as beams, and other two-dimensional structures (for example, see Section 6).

GEOROC has been developed on a VAX 11/750 using the VMS operating system and, to the extent possible, has been written in FORTRAN 77 so that it can be implemented on different computers and operating systems with a minimum of effort. Two notable exceptions occur that may need to be fixed if a computer other than a VAX is to be used. Several subroutines have names with more than six characters and there are some statements whose continuation exceeds 19 lines. The FORTRAN 77 compilers of most computers allow both these discrepencies to be rectified with options on the compilation commands.

3.2 CAPABILITIES AND LIMITATIONS

Perhaps the best way to describe GEOROC is by outlining the program's capabilities. The program is capable of solving problems involving the following conditions:

1. Geometry: Two dimensional (plane strain and plane stress) and axisymmetric.

2. Materials: Isotropic elastic and/or viscoelastic material properties. Each element in the mesh may have it's material specified individually, thus, permitting complex models to be onstructed such as stratigraphic sequences, etc.

3. Mining Sequence: Initial and sequential excavations and/or backfill can be simulated.

4. Pressure Loading: Pressure may be applied to the interior walls of the excavations for simulation of support systems, storage of fluids, backfill, etc.

5. Boundary Conditions: Kinematic (displacement) constraints, external surface loading, and skewed boundary orientation relative to global axes are incorporated.

6. Initial Stress State: Based on overburden specific weight and lateral earth pressure coefficient or based on body forces.

7. Factor of Safety: Based on creep strain (see also GNP, Section 4.2).

8. Alteration of Properties: Mechanical properties may be changed during the analysis; this alteration is done when a user-specified closure has been reached or at user-specified times and is intended for modeling backfill behaviour.

Several program limits are currently imposed because of array sizes:

Number of sequential excavations	$\leqslant 5$
Number of elements along the periphery of an excavation	$\leqslant 50$

Number of nodes at which output is requested	≤ 350
Number of elements at which output is requested	≤ 100
Number of time step maxima to be used	≤ 10

Except for these limits, GEOROC uses dynamic dimensioning to limit computer size requirements to the needs of a specific problem. Virtually all of the program storage is done in ARRAY(*). The initial dimension for this is set in the main program (GEOROC) and is currently 300,000. This may be changed by the user by applying the following formula:

The maximum dimension of ARRAY should be equal to or greater than:

$$
\begin{aligned}
&27\ (\text{No. nodes}) \\
&+ 169\ (\text{No. elements}) \\
&+ 79\ (\text{No. materials} + 1) \\
&+ 409 \\
&+ (\text{Frontwidth} + 3)[2 + 2(\text{No. nodes}) + (\text{Frontwidth} + 4)/2]
\end{aligned}
$$

A special mention should be made of GEOROC's capability to model backfill in potash or salt mines. This can be done in two ways, both of which are empirical. In the first, an internal pressure is applied to the room. As backfill consolidates it becomes stiffer and, hence, offers more resistance to closure. Based on laboratory data or other information, the user can specify the internal pressure on the room boundary and the times at which they are to be increased (until at some point they become equal to the lithostatic stress). For this approach it is usually necessary to know the rate of closure and, thus, a preliminary analysis with zero internal pressure is necessary. The second method is to assign the backfill elastic properties and to alter the properties at either specified times or when a specified amount of closure has occurred. The latter option allows the user to relate closure to laboratory determinations of the relationship between elastic properties and consolidation such as provided by Beddoes (1984) or Hansen (1976), for example.

GEOROC has the capability to simulate sequential excavations of rooms or drifts at user-specified times. GEOROC changes the material properties of the selected elements to those of air and adjusts the stress fields so that no normal or shear stresses occur at the boundary of the excavation. Instructions for specifying this option are given in Section 5.1 and an example case is described in Section 6.4.

3.3 MATHEMATICAL BASIS

The finite element method is used by GEOROC with eight-noded isoparametric quadrilateral elements. The formulation of continuum mechanics problems involves equations of equilibrium and compatibility with appropriate boundary and initial conditions. The constitutive relations describing the material behaviour are also incorporated into the formulation. The program uses the direct stiffness or displacement method. As outlined by Greenbaum and Rubinstein (1968) the potential energy for each element in the discretized structure being analyzed is minimized to result in a system of algebraic equations for each element in terms of the nodal displacements for the element and the applied forces acting on the element. Matrix superposition of the resulting elemental stiffness matrices and elemental load vectors yields a global system of governing equations for the entire structure being analyzed. The resulting set of linear algebraic equations is then modified to account for the specified displacement boundary conditions. For an elastic analysis, the system of linear equations is solved directly for the unknown nodal point displacements using the process of Gaussian elimination. The corresponding stresses and strains are then calculated for each element from the known displacements.

Solution of problems involving time-dependent deformation is accomplished using an incremental procedure which treats the creep strains as initial strains. This incremental approach to solving creep problems is based on the assumption that the length of time being considered in an analysis can be subdivided into sufficiently small time intervals such that the stress in any element can be assumed to be constant over each time interval or time step (e.g., Zienkiewicz, 1977). This assumption allows

nonlinear creep problems to be solved as a successive series of linear problems for each time step by treating the incremental creep strain, based on the average elemental stress over any time step, as the initial strain for the time step. These initial strains are included in the evaluation of the nodal displacements and elemental stresses and strains. All viscoelastic analyses are preceded by an initial calculation of the elastic stress distribution. The total elastic displacements are set to zero after this initial elastic solution such that the total displacements predicted on subsequent nonlinear creep calculations reflect only the total viscoelastic deformation.

The length of the time steps used in this incremental solution of nonlinear creep problems has a significant impact on the accuracy of predictions that are obtained. Although increased accuracy results from a decrease in the time step since the incremental change in stress over the time step approaches zero, the user should be aware of the associated increase in computing costs. To affect a cost effective viscoelastic analysis, GEOROC continually monitors the maximum change in elemental effective stress and makes appropriate adjustments to the time step to decrease the required number of time steps, while still maintaining an acceptable accuracy.

A material excavation process is modeled by modifying the system of equations to reflect any elements that are removed and the associated normal pressure or stress relief that occurs on the periphery of the specified evacuation or excavation. GEOROC implements this procedure by changing the relevant element material properties to those of air, rather than actually deleting the elements. The process is assumed to be instantaneous and the corresponding elastic response is determined. If a single excavation is requested, the corresponding elastic response is included in the initial elastic pass. If multiple processes are specified, the resulting incremental elastic response from subsequent processes is retained in the total deformation and stress history.

It is beyond the scope of this report to describe the detailed mathematical formulation on which GEOROC is based. For a more comprehensive review the reader is referred to Zienkiewicz (1977), Desai and Abel (1972), Greenbaum and Rubenstein (1968), and Owen and Hinton (1980). The latter

reference contains a particularly good description of non-linear finite element analysis, and many of the subroutines in GEOROC are adapted from that text. Those readers who are interested in the specific solution strategy used in GEOROC are referred to Callahan et al., (1985).

3.4 COMPUTER IMPLEMENTATION

GEOROC consists of one (short) main program and 52 subroutines and functions. A generalized flow chart showing the progression through the program is presented in Figure 3-1 and a brief description of each subroutine is provided in Appendix A. It is seen that the main program control is done from within subroutine GEOROC and that the main mechanics calculations are controlled by subroutine STRUCPROC. It is strongly recommended that any potential users of this program become familiar not only with Figure 3-1 and Appendix A but also with the actual program listing.

GEOROC requires one input file and produces one, or optionally, two output files. The input is expected at the standard input, (usually the keyboard), and the main output file goes to the standard output (again, this is usually the terminal), but these can be redirected to files by operating system commands (such as the ASSIGN command under VAX/VMS, or I/O redirection with UNIX). If the GNP database option is selected, it also produces a binary output file from FORTRAN unit #22, which on almost all systems means a file is produced called 'TAPE22', (or in the case of VAX/VMS, 'TAPE22.DAT') which contains the binary data (not readable by humans using text editors or print commands) for input to the GNP plotting programs.

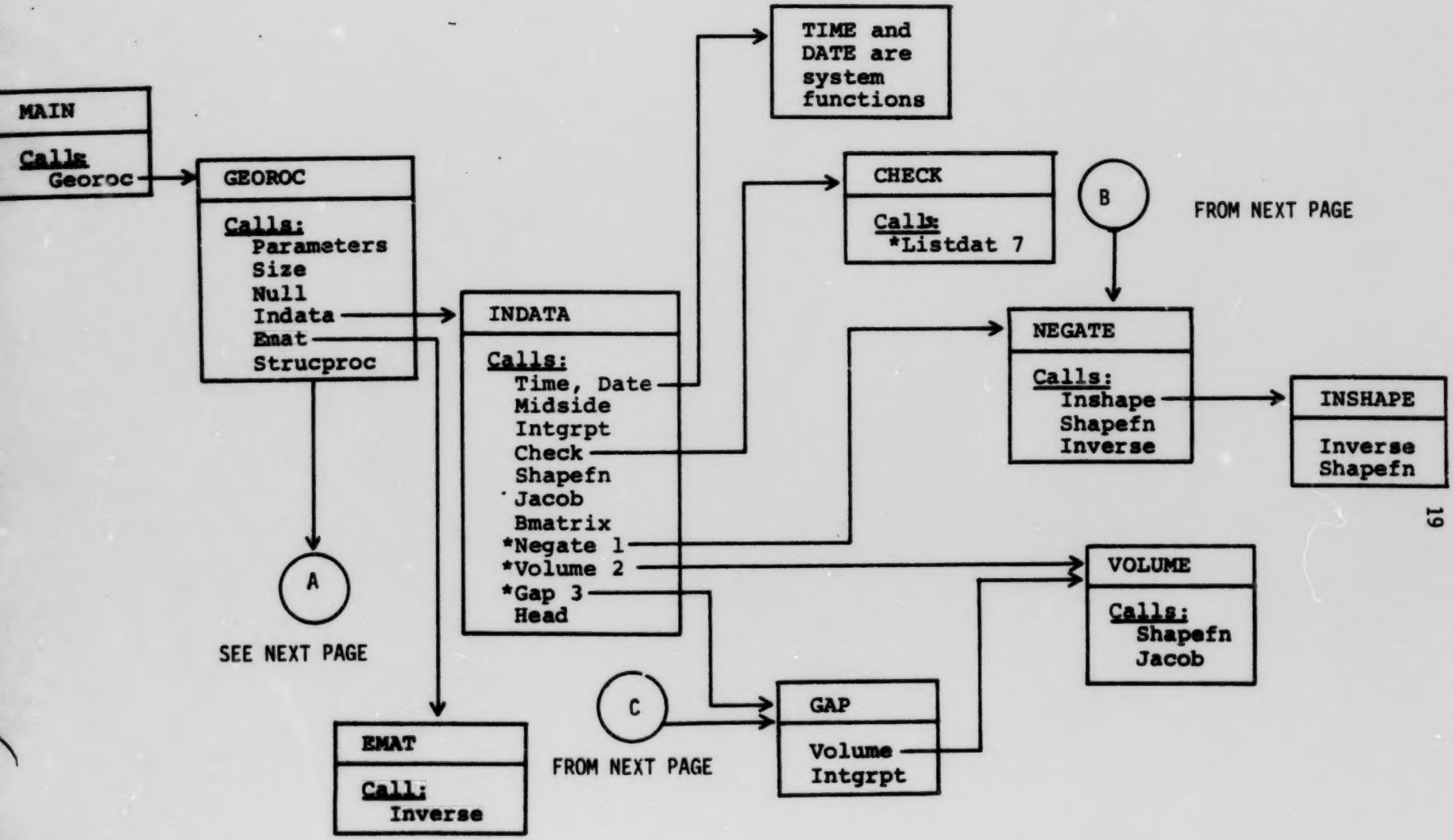


Figure 3-1. GEOROC Flowchart

*Indicates optional subroutines
Numbers refer to footnotes

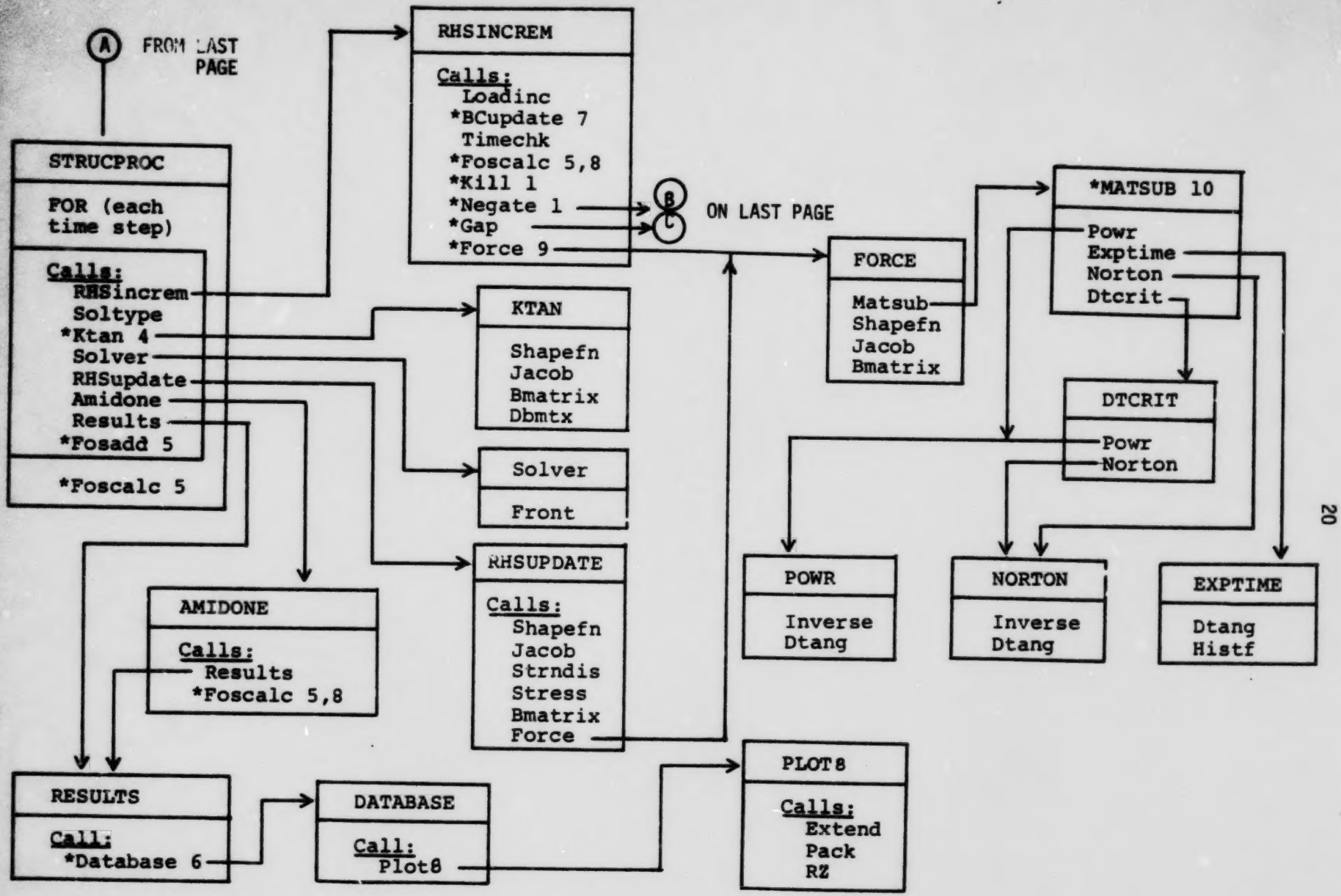


Figure 3-1. (continued) GEOROC Flowchart

FOOTNOTES for Figure 3-1.

1. KILL and NEGATE are called only when excavations occur, during the timestep in which it occurs. INDATA only calls NEGATE for excavations that occur at the starting time.
2. VOLUME is only called when a 'change of material properties' is called for and only then when the criterion for the change is that it occur at a particular volume change in the element, rather than at a specified time. VOLUME is then called by INDATA at the start, and by GAP, to compare with the first value, later.
3. GAP is only called when change of material properties (backfill) is requested. See (2) above.
4. KTAN is only called when there is a need for a change in the stiffness matrix. (After excavation, backfill, etc.)
5. Although FOSADD is called at each timestep, FOSADD and FOSCALC only perform a calculation when Factors of Safety are requested. (Flag
6. ISFOS on CARD 1).
   DATABASE is only called when the flag requesting a plotting
7. database, or 'GNP file' is set. (NDATA on CARD 1).
   LISTDAT is only called by CHECK if check finds an error on the input and aborts the program. LISTDAT just lists out the rest of
8. the input file that was not even examined.
   FOSCALC must always be called just before the program terminates, and thus it may be called from RHSINCREM or from AMIDONE, which each do tests to determine if the program has exceeded some limit,
9. and contain STOP statements.
   FORCE is only called by RHSINCREM if there has been no excavation or backfill on a given time step, i.e. neither NEGATE or GAP were
10. called by earlier in the same run of RHSINCREM.
    MATSUB calls just one of EXPTIME, NORTON, or POWR, depending on the constitutive law applying to a given element. DTCRIT is not called for elastic elements.

3.5 COORDINATES AND UNITS

Figure 3-2 shows the coordinate systems used by GEOROC for plane stress/strain and axi-symmetric analyses. X (or R) is taken as horizontal and the X-Y (or R-Z) plane is coincident with the plane of the page.

Although any system of units is allowed as long as the user is consistent, it is recommended that the SI system (M, s, kg) be used, but with MPa for stress rather than Pa. As noted in the user's guide (Section 5.1), this requires that the material property, density, be entered as 10^{-6} x density in kg/m^3. It should be noted that the creep strain factor of safety can only be obtained if the recommended system of units is used. (Should a user require the factor of safety while using a different set of units, then he will need to modify the coefficient of Equation 2-5 appropriately. This equation is implemented in subroutine FOSCALC.)

It may occasionally be convenient for a user to use units other than those recommended above. For example, in Section 6.4 an analysis is presented in which the time unit is in days and all other units are SI and MPa. In this case all input requiring time (except the gravitational constant which stays in m/s^2) is in days (e.g., times for output printing, times for excavations, etc.). In addition, some of the constitutive law parameters in Table 2-1 contain time and must also be converted appropriately. For example, in Table 2-1 the coefficient A for the power law is given in units of [MPa^{-m} s^{-n}]. To obtain A in days, it is necessary to multiply the A (in seconds) by $(86,400)^n$.

3.6 OUTPUT

GEOROC generates large quantities of data and the user has considerable control over how much will be printed as output. In the first segment of the output, GEOROC summarizes the input data used for the run. Although this feature is optional, it is recommended that the user always invoke it when doing 'production' runs. The user can also request that all Gauss point coordinates be listed. At each specified node, GEOROC provides the X and Y (or R and Z) displacements as well as velocities, that is

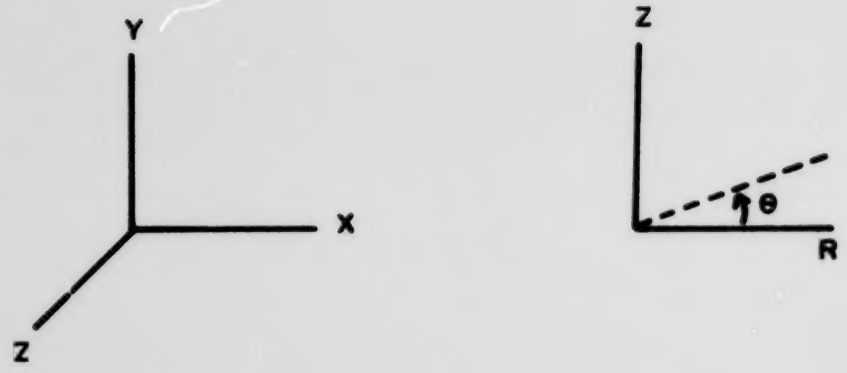


Figure 3-2 Coordinate Systems Used by Georoc for Plane Stress/Strain (a) and Axi-Symmetric (b) Analyses

displacements per time unit. The velocities are useful indicators of the rate of creep. At each specified element, GEOROC outputs at each of the four Gauss points the stress tensor, the principal stresses and their angle, the equivalent (or effective, or deviatoric) stress as well as the equivalent inelastic strain rate. Note that no units are provided in the output listing. The output is consistent with the input, for example, if the user has specified all input in the SI system (m, s, kg, Pa) then the displacements, velocities and stresses will be output in m, m/s and Pa respectively.

For a problem such as described in Section 6.4 where the mesh has 104 elements, 349 nodes and the analysis extends for 365 days, a very large amount of data can be generated. The user must be careful to request sufficient output so he can properly interpret the problem yet not be swamped by paper. The importance of using post-processing graphics, as described in Sections 4.2 and 4.3, as an aid to interpreting the output cannot be overstated.

4 PRE- AND POST-PROCESSORS

4.1 GEOMESH

A mesh describing the region to be analyzed must be prepared and entered as part of the input file for GEOROC. The mesh data includes the element numbers and their material type and connected node numbers as well as the node numbers and their coordinates. In simple cases this information may be calculated manually, but in practice, virtually all meshes need to be generated by computer. This task is performed using program GEOMESH and its output is merged with additional information to form a complete file for GEOROC.

GEOMESH is quite flexible and allows a variety of complex meshes to be generated with relative ease. It is based on the concept of mapping using isoparametric shape functions (Zienkiewicz, 1977, p.204). A separate user's manual has been written for this program and is included in Appendix B. Rather than repeat the description here, the reader is referred to Appendix B for details. GEOMESH is written in FORTRAN 77.

4.2 GNP

GNP is a mnemonic for Graphs and Plots. It is a set of three separate post-processor programs (ALGEBRA, TPLOT, SPLOT) which are used to help the user inspect the large mass of output data which is produced by GEOROC. The GNP programs access a common data base of stresses, displacements, nodal coordinates, times etc. which has been prepared by GEOROC. The data base is in the form of a binary, unformatted FORTRAN file and is output with the file name TAPE22. TPLOT generates x versus time plots, where x is an appropriate variable, such as a stress or displacement, selected by the user. SPLOT generates x versus distance plots. These plots may be produced on a line printer or on a graphics plotter. ALGEBRA is a program which allows the user to access the data base and create functions of these data. The function values are output in the plotting data base format so that they in turn may be plotted by TPLOT or SPLOT. ALGEBRA offers all the FORTRAN operations and

functions plus PMAX (maximum principal value of a tensor), PMIN (minimum principal value of a tensor), TMAG (magnitude of a tensor) and MCFS (Mohr Coulomb factor of safety).

The GNP programs rely on NCAR Graphics, (see Section 4.3) to output graphs on a plotter. NCAR is not required if only printer plots are being generated. Note that some options such as connecting dots, and superimposing more than one plot do not apply for plots made using a line printer.

It should be noted that all of the GNP subroutines are written in FORTRAN 77. The memory allocation routine is written in COMPASS, the CDC assembly language. If a blank common is large enough, this routine is not called.

A detailed description of how to use the GNP programs as well as illustratives examples are provided in Section 5.

4.3 NCAR GRAPHICS

NCAR Graphics is a non-proprietory graphics package which has been developed by the U.S. National Center for Atmospheric Research (NCAR, 1983). It is used by GNP and no input or instructions are required from the user. NCAR takes the metacode which has been generated by the GNP software and produces the hardware commands necessary to generate the requested plots. This software is computer and plotter dependent and the version supplied with this project is for a VAX computer with a VMS operating system. Anyone wishing to obtain GEOROC and the post-processors would need to make their own arrangements for purchasing the NCAR software which is compatible with their computing system. The graphics devices which are supported by NCAR are listed in Table 4-1.

It should be noted that the NCAR package contains a large number of files, some of which can perform some very sophisticated graphics tasks. However, only the NCARPLOT directory and one of its subdirectories, TRANS, are necessary for the GNP post-processors.

Table 4-1. A List of Graphics Devices Supported by NCAR.

1. Tektronix 4006 Graphics terminals.
2. Tektronix 4014 Graphics terminals.
3. Tektronix 4025 Raster Graphics terminals.
4. Lear Siegler ADM-3 dumb terminal with Digital Engineering Retrographics board.
5. DEC VT100 terminal with Digital Engineering Retrographics board.
6. Printronix P-300 dot matrix printer plotters (sold by DEC as LXY11).
7. Trilog C-100 color dot matrix printer plotters.
8. Integral Data Systems IDS 560 Paper Tiger printer plotter.
9. Chromatics 7900 color graphics device.
10. Hewlett-Packard 7475A pen plotter.
11. DEC GIGI terminal.
12. Grinnell graphics device (software from Grinnell required).

5 USER'S GUIDE

5.1 GEOROC

This section provides detailed instructions for the user on how to prepare the input file for program GEOROC. User instructions for program GEOMESH can be found in Appendix B and instructions for the GNP programs are outlined in Section 5.2.

All program input option requests are through 80-column 'card images' in a single input file. The number and sequence of cards input are affected by the options selected. The following input description will contain a separate description for each distinct card image, indicating the meanings of all fields. For those affected, a note at the bottom will indicate the total number of occurrences of a card or group of cards, and whether a following card or group is to be skipped, owing to not selecting the option that they affect. Those cards that might be skipped will have square brackets around the card numbers, and cards that must always be present will not, so that the manual may be read through during input preparation as if it were a 'flowchart' of the input process.

	NAME	FIELD DESCRIPTION	COLUMNS (FORMAT)
CARD 1	NPOIN	Number of nodal points in total mesh	1-5 (I5)
	NELEM	Number of elements in total mesh	6-10 (I5)
	NVFIX	Number of restrained nodes	11-15 (I5)
	NTYPE	Type of geometric model used: 1 = Plane Stress 2 = Plane Strain 3 = Axisymmetric	16-20 (I5)
	NMATS	Number of different material types, (see cards 11,12)	21-25 (I5)
	NPRBT	Problem type code: 1 = Time Dependent (Creep) 2 = Static (Elastic Only)	26-30 (I5)
	NDATA	Prepare GNP plotting database (0=No, 1=Yes)	31-35 (I5)
	NCHBC	Number of boundary conditions changed after the initial elastic pass. (Generally 0).	36-40 (I5)
	NELGP	Number of elements whose material properties change at some point (i.e. are backfilled)	41-45 (I5)
	MFRON	Maximum front width	46-50 (I5)
CARD 2	TITLE	May be any 80 character string. Printed at top of each page of output.	1-80 (A80)
CARD 3	IAMDN	Execution flag: 0= Problem will be solved 1= Data checked only	1-5 (I5)

DECISION: Cards 4,5,6 relate to time steps for creep problems.
If problem is elastic, (NPRBT of CARD 1 = 2) go to CARD 7

	NAME	FIELD DESCRIPTION	COLUMNS	(FORMAT)
[CARD 4]	NINCS	Number of time steps allowed for creep solution, i.e. program stops after this number of steps whether time limit has been reached or not. Typically about 300 for 1 year of analysis.	1-5	(I5)
	MXDTS	Number of time step maxima to be imposed. This is generally 1. More than one is used when it is desired to, say, permit time steps of up to 10 seconds for the first hour, of up to 60 seconds for the next 23 hours, 1000 seconds for the next 2400 hours, etc.	6-10	(I5)
[CARD 5]	FTIME	Final time value, at which program halts.	1-10	(F10.0)
	TUNIT	Time unit, with which all time entries and related coefficients in the input must be consistent. SECS = Times are in seconds DAYS = Times are in days YEARS = Times are in years (365.25 days)	11-15	(A5)
[CARD 6]	TTMXS	Time at which this time step maximum comes into effect. (First one should be 0.0)	1-10	(F10.0)
	DTMXS	Maximum size time step allowed.	11-20	(F10.0)
REPETITIONS:		The total number of occurrences of CARD 6 must be the MXDTS value on CARD 4.		

	NAME	FIELD DESCRIPTION	COLUMNS (FORMAT)
CARD 7	NPRNT	'Print node and element values' flag 0 = No, do not print them, 1 = Yes, print them. The same code applies to 3 flags below.	1-5 (I5)
	NGPCD	'Print Gauss Point coordinates' flag	6-10 (I5)
	IPRFO	'Print total external nodal forces' flag	11-15 (I5)
	IELAS	'Print elastic results' flag	16-20 (I5)
	KOUTP	Print option code: 1 = Print nodal displacements and residual forces only. 2 = Print reactions at restrained boundary nodes only. 3 = Print Gauss point stresses only 12, 13, 23, 123 = combinations of the above.	21-25 (I5)
	KPRNT	Frequency of printing option code: 0 = Elastic run, therefore only 1 output 1 = Output at intervals of time steps 2 = Output at specified times 3 = Both (1) and (2) cause output	26-30 (I5)
	NOUTP	Print interval, i.e. number of time steps between outputs if KPRNT = 1 or 3. Must be 0 if KPRNT = 2.	31-35 (I5)
	NTIME	Number of specified times at which output will occur if KPRNT = 2 or 3. Must be 0 if KPRNT = 0 or 1.	36-40 (I5)
	NPRND	The number of nodes for which displacements and forces will be output.	41-45 (I5)
	NPRNS	The number of elements for which Gauss point stresses will be output. NOTE: For both 'NPRND' and 'NPRNS' above, an input of zero or blanks causes ALL of the mesh to be included in the output.	46-50 (I5)
	ISFOS	'Calculate and print the creep strain safety function (Equation 2-5)' flag. (0 = No, 1 = Yes). Note that this option can only be used if the input data is in SI units using MPa.	51-55 (I5)

DECISION: If outputs are not based upon specified times, i.e. NTIME = 0 then go to CARD 9

	NAME	FIELD DESCRIPTION	COLUMNS (FORMAT)
[CARD 8]	PTIME(1)	Time value at first desired output.	1-10 (F10.0)
	PTIME(2)	Time value at second desired output.	11-20 (F10.0)
	.		
	.		
	.		
	.		
	PTIME(NTIME)	Time at last desired output.	

REPETITION: This array is of length NTIME, so if NTIME>8 then CARD 8 will run to more than one card, to (NTIME/8 rounded up) cards total, each 10F10.0 format.

DECISION: If the default of 'all nodes to be output' (NPRND of CARD 7 = 0) was chosen, skip to CARD 10

[CARD 9]	NOUTD(1)	The first nodal point number to appear in the output.	1-5 (I5)
	NOUTD(2)	The second nodal point number.	6-10 (I5)
	.		
	.		
	.		
	.		
	NOUTD(NPRND)	The last nodal point number output.	(I5)

REPETITION: As above, this is an array, entered onto 80 column 'card images' at 16I5 entries per card, and (NPRND/16 rounded up) CARD 9 entries will appear.

DECISION: If the default of 'all elements to be output' (NPRNS of CARD 7 = 0) was chosen, skip to CARD 11.

[CARD 10]	NOUTS(1)	The first element number to appear in each output.	1-05 (I5)
	.		
	.		
	.		
	.		
	NOUTS(NPRNS)	The last element number output.	(I5)

REPETITION: As above, CARD 10 will necessarily appear a total of (NPRNS/16 rounded up) times, each with format 16I5.

CARD GROUP: CARDS 11 and 12 describe a material type, so at least one appearance of them is mandatory. The group is repeated for each material.

	NAME	FIELD DESCRIPTION	COLUMNS (FORMAT)
CARD 11	NUMAT	The material type number.	1-5
	NCRIT	The constitutive law type code: 0 = Elastic 1 = Time Power Law 2 = Exponential Time Law 3 = Norton Power Law	6-10
CARD 12	PROPS(1)	Young's modulus. Units (force/distance**2) must be consistent with forces used in loads and distances in nodal coordinates. Exponential input (i.e. 205.3E+09) is allowed.	1-10 (F10.0)
	PROPS(2)	Poisson's ratio.	11-20 (F10.0)
	PROPS(3)	Element Thickness. Only need input non-zero if (NTYPE of CARD 1) = 1, i.e. it is a PLANE STRESS type problem. Then use same distance units as for nodal coordinates.	21-30 (F10.0)
	PROPS(4)	Density. Only non-zero if gravity loading (see CARD 17, CARD 21) is used. Then use (mass/dist**3) units that are consistent with distances of nodal coords, and mass unit * (gravity specified in CARD 21) = force units used in loads. If distance is metres and stresses are in MPa, then density should be in (millionths of kg)/m^3.	31-40 (F10.0)
FIELDS BELOW NOT REQUIRED FOR ELASTIC MATERIALS			
	PROPS(5)	Leading coefficient. All of the creep laws have a basic multiplier value.	41-50 (G10.0)
	PROPS(6)	Power applied to stress.	51-60 (G10.0)
	PROPS(7)	If using the Time Power Law: Power on time. If using Exponential Time Law: e_a. Otherwise leave blank.	61-70 (G10.0)

FIELDS BELOW ONLY REQUIRED FOR EXPONENTIAL TIME LAW

PROPS(8)	The constant B	71-80 (G10.0)
PROPS(9)	The constant ė* (This is the first field on an additional CARD 12 required for exponential time law materials).	1-10 (G10.0)

REPETITION: If you have more than one material type in your problem, go back to CARD 11, and repeat CARDS 11 and 12 for the second material, and so on, for as many materials as required. (Total number of materials must appear on CARD 1). Otherwise, continue to CARD 13.

	NAME	FIELD DESCRIPTION	COLUMNS (FORMAT)
CARD 13	NUMEL	Element Number of Element being described.	1-5 (I5)
	MATNO	Material Number (See CARD 11) for this element.	6-10 (I5)
	LNODS(1)	'Bottom Left' node (smallest x and y) of an element.	11-15 (I5)
	LNODS(2)	Next node, proceeding around element counterclockwise.	16-20 (I5)
	.		
	.		
	.		
	.		
	LNODS(8)	Last node number before coming to lower left.	46-50 (I5)

REPETITION: There must be one CARD 13 for each element in the mesh, so there are (NELEM) occurences in total of CARD 13.

	NAME	FIELD DESCRIPTION	COLUMNS (FORMAT)
CARD 14	IPOIN	Node number of node being described.	1-5 (I5)
	COORD(IPOIN,1)	X (or R) coordinate of node.	6-20 (F15.0)
	COORD(IPOIN,2)	Y (or Z) coordinate of node.	21-35 (F15.0)
REPETITION:		There must be one CARD 14 for each node in the mesh, so there are (NPOIN) occurences in total of CARD 14.	
CARD 15	NOFIX	Number of node that has a prescribed restraint.	1-5 (I5)
	IFPRE	Code describing movement restraints on node: 10 = No movement in X (or R) direction. 01 = No movement in Y (or Z) direction. 11 = No movement in either of above directions. 00 = No restraint to start with, but this node will have more boundary conditions added after elastic pass. (This can only occur if (NCHBC of CARD 1) 0.	6-10 (I5)
	PRESC(1)	Value of prescribed displacement, X (or R).	11-20 (F10.0)
	PRESC(2)	Value of prescribed displacement, Y (or Z).	21-30 (F10.0)
	PRESC(3)	Value of prescribed displacement, Z (or T).	31-40 (F10.0)
REPETITION:		CARD 15 must appear once for each restrained node in the mesh, so (NVFIX of CARD 1) occurences of it must appear.	

NOTE: All of the occurrences of CARDS 13, 14 and 15 are normally generated as an output file by the 'mesh generator' GEOMESH.

DECISION: CARD 16 relates to those nodes that change their boundary conditions after the elastic pass, so if (NCHBC of CARD 1) is zero, then skip CARD 16, go to CARD 17.

	NAME	FIELD DESCRIPTION	COLUMNS (FORMAT)
[CARD 16]		Has exactly the same fields as CARD 15, but these boundary conditions and prescribed displacements will not be imposed until after the elastic pass.	
CARD 17	NPLOD	Number of point loads applied to the model.	1-5 (I5)
	NEDGE	Number of edge loads applied to the model.	6-10 (I5)
	IGRAV	Flag(1=yes, 0=no) for gravity loads on model.	11-15 (I5)
	ISTRS	Flag(1=yes, 0=no) for initial stress loads.	16-20 (I5)

NOTE: Generally, only one of IGRAV or ISTRS is set.

Note also that CARD 17 is similar to CARD 1 in that it indicates whether entire groups of cards below will be input at all. These flags will be referred to in four DECISIONS below.

DECISION: If NPLOD is zero, skip CARD 18 and continue on to DECISION above CARD 19. Otherwise, there will be NPLOD appearances of CARD 18.

[CARD 18]	LODPT	Point number of nodal point to which this point load is applied.	1-5 (I5)
	POINT(1)	Load applied along X (or R) axis	6-15 (F10.3)
	POINT(2)	Load applied along Y (or Z) axis	16-25 (F10.3)

REPETITION: There are additional occurences of CARD 18 until each point load (NPLOD of them) applied to the model is quantified.

DECISION: If NEDGE of CARD 17 = 0 skip CARD 19 and CARD 20 and go to CARD 21.

	NAME	FIELD DESCRIPTION	COLUMNS (FORMAT)
[CARD 19]	NEASS	Element number of the element whose edge load is described by this CARD 19,20 pair.	1-5 (I5)
	NOPRS(1)	Node numbers of loaded edge of element number NEASS going counterclockwise around the element.	6-10 (I5)
	NOPRS(2)		11-15 (I5)
	NOPRS(3)		16-20 (I5)
[CARD 20]	PRESS(1,1)	Normal load applied to NOPRS(1), above.	1-10 (F10.0)
	PRESS(1,2)	Tangent load applied to NOPRS(1), above.	11-20 (F10.0)
	PRESS(2,1)	Normal load applied to NOPRS(2), above.	21-30 (F10.0)
	PRESS(2,2)	Tangent load applied to NOPRS(2), above.	31-40 (F10.0)
	PRESS(3,1)	Normal load applied to NOPRS(3), above.	41-50 (F10.0)
	PRESS(3,2)	Tangent load applied to NOPRS(3), above.	51-60 (F10.0)

REPETITION: The card group 19,20 must be repeated until one occurrence of the pair for each of NEDGE edge loads has appeared.

DECISION: If both IGRAV and ISTRS of CARD 17 are zero, then skip over CARD 21 and go to the DECISION for CARD 22,23,24.

[CARD 21]	THETA	Angle that gravity makes with positive Y (or, for axisymmetric problems, Z) axis. Where axes are aligned with true vertical, this is just 0.0.	1-10 (F10.0)
	GRAVY	Gravitational Acceleration (dist/time**2) Distance units should be the same as those for nodal coordinates, but TIME IS ALWAYS IN SECONDS. Thus 9.81 and 32.2 should be almost always the only values required.	11-20 (F10.0)

DECISION: If ISTRS = 0 on CARD 17, skip CARD 22,23,24 and go to CARD 25.

	NAME	FIELD DESCRIPTION	COLUMNS (FORMAT)
[CARD 22]	NLAYS	Number of layers for different initial stress specification. Must be at least 1.	1-5 (I5)
	OBSTR	Overburden stress(force/dist**2) applied to the top of the model. (SIGN CONVENTION: Compressive Stress is NEGATIVE).	6-10 (I5)

DECISION: If NTYPE=1 on CARD 1, then input CARD 23 and DO NOT input CARD 24. If NTYPE >1, skip CARD 23 and go to CARD 24.

[CARD 23]	YYTOP	Top elevation of this layer.	1-10 (F10.0)
	YYBOT	Bottom elevation of this layer.	11-20 (F10.0)
	AVDEN	Average density of the mesh.	21-30 (F10.0)
	XLESX	In plane horizontal stress coefficient.	31-40 (F10.0)

DECISION: If NTYPE > 1 on CARD 1, then input CARD 24, instead of CARD 23.

[CARD 24]	YYTOP	Top elevation of this layer.	1-10 (F10.0)
	YYBOT	Bottom elevation of this layer.	11-20 (F10.0)
	AVDEN	Average density of the mesh.	21-30 (F10.0)
	XLESX	In plane horizontal stress coefficient.	31-40 (F10.0)
	ZLESZ	Out of plane horizontal stress coefficient.	41-50 (F10.0)

REPETITION: Either CARD 23 OR CARD 24, not both, is to occur NLAYS (of CARD 22) times.

	NAME	FIELD DESCRIPTION	COLUMNS (FORMAT)
CARD 25	NUMEX	Number of excavations (Note: a number of elements all removed at the same point in time count as a single excavation).	1-5 (I5)
DECISION:	If NUMEX=0 above, it should be the last card in the input. It is still possible to go to CARD 30 and call for change of element properties, but as this is normally used only to 'backfill' excavated elements, a zero for CARD 25 would normally be the end.		
[CARD 26]	NEREM	Number of elements removed.	1-5 (I5)
	NEPEX	Number of elements on the excavated (free) surface, going all around it.	6-10 (I5)
	TMEXC	Time at which this excavation occurs.	11-20 (I5)
DECISION:	If NEREM, above, is zero, skip CARD 27, go to CARD 28.		
[CARD 27]	LEREM(1)	Element Number of first removed element.	1-5 (I5)
	LEREM(NEREM)	Element Number of NEREM-th removal	All fields (I5)
[CARD 28]	LEPEX	Element number of an element on the free excavated surface.	1-05 (I5)
	NOEXS(1)	These are the nodal point numbers of the three nodes of LEPEX that are on the excavated surface. They MUST occur on this card in a particular order, GOING AROUND ELEMENT NUMBER LEPEX COUNTERCLOCKWISE.	5-10 (I5)
	NOEXS(2)		11-15 (I5)
	NOEXS(3)		16-20 (I5)
[CARD 29]	PNORM(1)	Normal stress on excavated surface at node NOEXS(1).	1-10 (F10.0)
	PSHER(1)	Shear stress on excavated surface at node NOEXS(1).	11-20 (F10.0)
	PNORM(2)		21-30 (F10.0)
	PSHER(3)	Same as above for NOEXS(2)	31-40 (F10.0)
	PNORM(3)		41-50 (F10.0)
	PSHER(3)	Same as above for NOEXS(3)	51-60 (F10.0)
REPETITION:	The card pair 28,29 must ocur a total of NEPEX times below CARD 27, to describe all elements on the excavated surface.		

REPETITION: This is the only place in the input where there are "nested DO loops", i.e. two levels of repetition. The card group (CARD 26, CARD 27, NEPEX * (CARD 28, CARD 29)) must as a group occur (NUMEX of CARD 25) times in total, to describe all excavations. In sum, a description of the excavation section of the input, for NUMEX > 0, is: CARD 25, NUMEX * (CARD 26, CARD 27, NEPEX * (CARD 28, CARD 29)) where of course the NEPEX value on CARD 26 may change each time.

DECISION: This is the end of the input file, unless NELGP of CARD 1 > 0.

	NAME	FIELD DESCRIPTION	COLUMNS (FORMAT)
[CARD 30]	RTOLG	Volume ratio of excavated element, current Vol/Initial Vol, when property change (backfill) will occur.	1-10 (F10.0)

DECISION: If backfill is to be made when the excavation has reached a set value of volume closure, RTOLG above will be non-zero, and input will now be NELGP occurences of CARD 31. If backfill is to occur at user specified times, input RTOLG as 0.0 above, and the remaining input is NELGP occurrences of CARD 32 instead.

[CARD 31]	MELGP	Element number whose material properties are to change.	1-5 (I5)
	MATGP	Material number of the new material type.	6-10 (I5)

REPETITION: There are NELGP occurences of CARD 31, and that is the end of the file.

[CARD 32]	MELGP		1-5 (I5)
	MATGP	Both of these are as above on CARD 31.	6-10 (I5)
	BKTIM	The time at which the property change, or backfill, for element MELGP occurs. NOTE: If a number of elements are all backfilled at once, they must occupy consecutive repetitions of CARD 32, and all repetitions of CARD 32 must be in sorted order of increasing time of backfill.	11-20 (F10.0)

REPETITION: If the time is used, there are NELGP occurences of CARD 32 and that is the end of the file.

5.2 GNP USER GUIDE

5.2.1 ALGEBRA

Program ALGEBRA is one of three GNP programs which allows the user to post-process, i.e. inspect and assess, the large volume of output data produce by GEOROC. ALGEBRA allows the user to form functions of GEOROC output data. When ALGEBRA is run it accesses two input files which must have the names ALGEBRA-BIN-INPUT and ALGEBRA-INPUT. Its output file is called ALGEBRA-OUTPUT. A schematic representation of the operation of ALGEBRA and its relationship to GEOROC and the other GNP programs is shown

in Figure 5-1 ALGEBRA-BIN-INPUT is the binary data base which GEOROC produces. Since the file produced by GEOROC is automatically named TAPE22, it is necessary for the user to change the name to ALGEBRA-BIN-INPUT, if he wishes to do an ALGEBRA run. The arrays stored in the data base (TAPE22) for each output time specified by the user are listed in Table 5-1.

Table 5-1

Coordinate Data	Nodal Data	Element Data	Global Data
X	DISPLX	SIGX	(NONE)
Y	DISPLY	SIGY	
		TAUXY	
		SIGZ	
		SIGEFF	

X is X-coordinate, DISPLX is the displacement in the X-direction. SIGX is the (xx) stress component and so on. SIGEFF is the effective, or equivalent, stress. It should be noted that the above variable names will be used, even if an axisymmetric (i.e. R-Z coordinates) analysis is done by GEOROC. In this case R corresponds to X and Z to Y. TAPE22 is created by GEOROC in subroutine PLOT8. Any user who is interested in the format and structure of this file is referred directly to PLOT8.

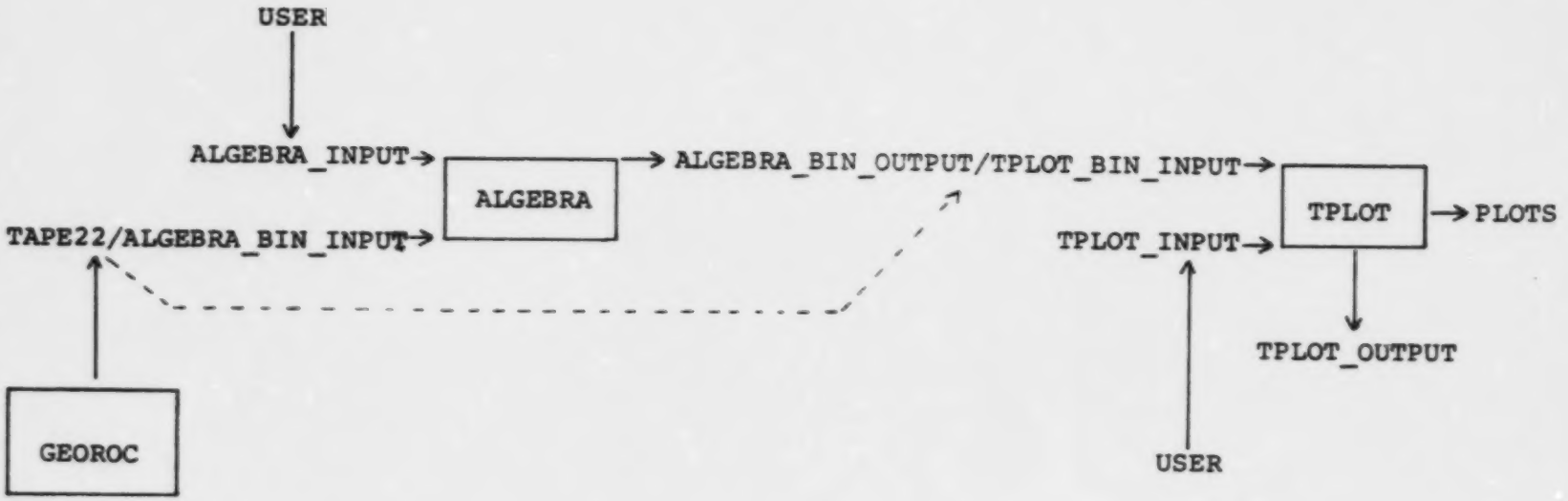


Figure 5-1. Schematic Representation of INPUT and OUTPUT Files Used by GNP. "/" Indicates that a File must be Renamed.

ALGEBRA-INPUT is the name of the file which the user must create and which instructs ALGEBRA regarding what new functions it is to create from the variables which are in TAPE22 (see Table 5-1).

Nodal, elemental, or global (scalar values representative of the system as a whole, e.g. total energy) functions can be created by the user by writing equations which define the new functions. The following rules must be used:

- The left hand side must contain only the new variable name and must be from 1 to 10 alphanumeric characters long, with a letter as its first character.

- A maximum of 20 equations is allowed.

- To include a sign with a variable or constant, parentheses must be used (i.e 6 + (-5)).

- The user must be consistent with his use of variable types (i.e. nodal, elemental or global). That is, the new variable will be of the same type as the variable on the right hand side. Elemental and nodal variables cannot be mixed.

- Quantities may be evaluated at specific nodes or elements by appending a "$" and the appropriate number e.g. DISPLX$113 refers to the X displacement at node 113. Elemental values are available at any of the four Gauss points within an element. The appropriate Gauss point and element is selected by using the formula (element - 1) * 4 + (Gauss point). For example, SIGX$147 refers to Gauss point 3 in element 37.

- All blanks, including blank cards, are ignored and may be used to improve legibility.

- The last input line is an END card.

The following FORTRAN functions are available: SIN, COS, SQRT, LOG, TAN, ASIN, ACOS, ATAN and EXP. Both integer and rea constants may be used and the exponential format (i.e. 5.23E6) is allowed.

Four special functions have been included in ALGEBRA which are commonly used in mechanical analyses:

PMAX (T_{11}, T_{22}, T_{33}, T_{12}, T_{23}, T_{13}) : calculates maximum principal value of tensor T.

PMIN (T_{11}, T_{22}, T_{33}, T_{12}, T_{23}, T_{13}) : calculates minimum principal value of tensor T.

TMAG (T_{11}, T_{22}, T_{33}, T_{12}, T_{23}, T_{13}) : calculates the magnitude of the tensor T from the formula.

$$TMAG = [\ (T_{11}-T_{22})^2 + (T_{11}-T_{33})^2 + 6(T_{12}^2 + T_{23} + T_{13}^2)\ ]^{1/2}$$

All six parameters must be included in the above function. For a two-dimensional tensor T_{33}, T_{23} and T_{13} must be set to zero.

The MCFS function calculates a Mohr-Coulomb safety factor from a tensor T. The user must supply a cohesion, S, and an angle of internal friction, P (in degrees) e.g.

$$FS = MCFS\ (S,P,T_{11},T_{22},T_{33},T_{12})$$

The following is an example of an ALGEBRA-INPUT file which calculates the fractional vertical and horizontal closure of an underground room with height 3.3 m and width 10.0 m. It also calculates the Mohr-Coulomb factor of safety assuming a cohesion of 7.5 MPa and an angle of internal function of 25 degrees.

```
V CLOSURE = (DISPLY $ 131 - DISPLY $ 252) /  3.3
H CLOSURE = (DISPLX $ 179 - DISPLX $ 339) / 10.0
FS        = MCFS (7.5, 25., SIGX, SIGY, SIGZ, TAUXY)
END
```

ALGEBRA-OUTPUT is the output file produced by ALGEBRA. It is structured with the same format as TAPE22 (See Table 5-1) and can be used as the input file for TPLOT and SPLOT.

If ALGEBRA discovers an error, the program stops and a diagnostic error message is printed.

Should further information regarding ALGEBRA be required, the user is referred to RE/SPEC (1982).

5.2.2 TPLOT

As explained in Section 4.2, TPLOT is one of three GNP programs which allow the user to post-process output data from GEOROC. TPLOT generates plots of a variable versus time or a variable versus a variable. It has automatic scaling and multiple curves may be drawn on one plot. The plots may be generted on a line printer or a graphics plotter.

The input and output files for TPLOT and their relationship to ALGEBRA and GEOROC are illustrated in Figure 5-1.

The file TPLOT_INPUT must be created by the user and contains the instructions for TPLOT. Two kinds of input are contained in TPLOT_INPUT. The first kind specifies names of variables from the data base in TPLOT_BIN_INPUT which are to be plotted. The second kind specifies the options which control how the plot(s) is (are) to be constructed. The input file is based on keywords and free field format. The following rules apply:

- Each line must contain 80 characters or less.
- All blanks, including blank cards, are ignored and may be used to improve legibility.
- All variables must be separated by a comma.
- A "*" is simultaneously a separator and a continuation character meaning further information is available on the next line.
- A "$" denotes a comment. All characters on a line to the right of the "$" will be ignored.
- The first variable on a line is taken as the keyword.
- All parameters are considered to be floating point variables.

A list of key words is given in Table 5-2 along with a description of their function.

Below is an example file which generates two plots at the same scale. The first plots the vertical displacement of node 131 (the roof of an underground room) versus time on the line printer while the second plots strss in a pillar (element 83, Gauss point 4, thus element number = 4 * (83-1) + 4 = 376 - see end of 5.2.1 for formula) versus time on the plotter. The time range for both plots is 10^3 to 10^6 seconds.

```
DISPLY,131
TMIN, 1.0E03
TMAX, 1.0E06
POINTS
PRINTER
END
SIGY, 376
POINTS
PRINTER
SAMESCALE
ENDALLDATA
```

Further details are provided in RE/SPEC (1982). A specific example of how ALGEBRA and TPLOT are used in supplied in Section 6.4.

5.2.3 SPLOT

Program SPLOT is one of three GNP programs used to post-process the output data produced by GEOROC. It generates plots of variables versus distance along a user-specified string of nodal points or elements. The input and output files associated with SPLOT are similar to those shown in Figure 5-1 for TPLOT except that the word TPLOT is replaced by SPLOT.

The preparation of input file SPLOT-INPUT by the user is similar to that for TPLOT-INPUT described in Section 5.2.2 and it is not necessary to repeat most of the material.

The keywords used by SPLOT-INPUT are described in Table 5-3. An example SPLOT-INPUT file is shown below. It generates one plot of vertical stress values versus horizontal distance at time 3.12E5 and another plot for time = 7.25E7. Output is on a line printer. Note that the stress values are for the first Gauss point of each specified element. The elements have been selected to provide the desired profile. The elements plotted are 83, 81, 59, 45, 12, 10, and to each has been applied the formula: element number = (element - 1) * 4 + 1.

```
$ PLOTS OF SIGY VS. HORIZ. DISTANCE
SIGY, 329, 321, 233, 221, 177, 45, 37
TIMES, 3.12E5, 7.25E7
POINTS
PRINTER
ENDALLDATA
```

Table 5-2. Keywords Used in TPLOT-INPUT

Keyword	Numeric Parameters	Description
DELTIME	Δt	Plot time increment. The default is every time step on the data tape. The program does no interpolation but takes the next time step $\geq$ the augmented time.
TMIN	t_{min}	The minimum time at which plot data will be read from plotting data base. The default is the first time step in the data base.
TMAX	t_{max}	The maximum time at which plot data will be read from the plotting data base. The default is the last time step in the data base.
FRAMESIZE	n	$1 \leq n \leq 3$ Three separate plot sizes may be selected. The default size is n=1. n=1, Square Format H/W=1/1. n=2, Vertical Format H/W=11/8. n=3, Horizontal Format H/W=8/11.
GRID		Grid lines will be drawn on the plot. The default is tick marks at intervals along each axis.
OVERLAY		All plots within a data set will be on the same frame. The default is one plot per frame.
POINTS		Points are plotted using the symbol (*) and are not connected by a straight line. The default is a line plot.
PRINTER		Line printer plots are output. The "OVERLAY" is not possible with printer plots.
SAMESCALE		All plots will have the same scale. The default is self-scaling for each plot.

Table 5-2 (Continued)

XSCALE	x_{min}, x_{max}	The minimum and maximum abscissa values are input. The defaults are the x_{min} and x_{max} values found by a scan of the data.
YSCALE	y_{min}, y_{max}	The minimum and maximum ordinate values are input. The defaults are the y_{min} and y_{max} values found by a scan of the data.
XLOG	n	$0 \leq n \leq 1$ The x-axis may be either linear or logarithmic. n=0, x-axis is linear. n=1, x-axis is logarithmic. The default is linear, n=0. Note: Zero and negative values in the data should be eliminated with XSCALE.
YLOG	n	$0 \leq n \leq 1$ The y-axis may be either linear or logarithmic. n=0, y-axis is linear. n=1, y-axis is logarithmic. The default is linear, n=0. Note: Zero and negative values in the date should be eliminated with YSCALE.
* XYPLOT		The next two cards input will be plotted against each other. Card #1 will be plotted along the x-axis and card #2 will be plotted along the y-axis. The default will have TIME along the x-axis.
* TITLE		The next card to be input may contain 80 characters of alphanumeric data to be written at the top of each plot. IF this key word is omitted, the title in the data base TPLOT-BIN-INPUT will be used.
END, ENDDATA		Signals the end of one data set. END and ENDDATA are identical.
ENDALLDATA		Signals the end of a job. An end-of-file also signals the end of a job.

* The position of this card or the following cards is important within a data set.

Table 5-3. Keywords Used in SPLOT-INPUT

Keyword	Numeric Parameters	Description
FRAMESIZE	n	$1 \leq n \leq 3$ Three separate plot sizes may be selected. The default size is n=1. n=1, Square Format HoW=1/1. n=2, Vertical Format H/W=11/8. n=3, Horizontal Format H/W=8/11.
GRID		Grid lines will be drawn on the plot. The default is tick marks at intervals along each axis.
OVERLAY		All plots within a data set will be on the same frame. The default is one plot per frame.
POINTS		Points are plotted using the symbol (*) and are not connected by a straight line. The default is a line plot.
PRINTER		Line printer plots are output. The "OVERLAY" is not possible with printer plots.
TMAX	t_{max}	The maximum time at which plot data will be read from the data tape. The default is the last time step on the tape.
TMIN	t_{min}	The minimum time at which plot data will be read from tape. The default is the first time step on the data tape.
TIMES	$t_1, t_2, t_3, .., t_n$	Array of times at which plots are requested.
ALLTIMES		Plot all times from TMIN to TMAX.
NTIMES	N	N evenly spaced plots from TMIN to TMAX. Default N=10.
DELTIME	Δt	Plots at every Δt increment between TMIN and TMAX.

6 VERIFICATION AND EXAMPLE PROBLEMS

GEOROC is a large and complex program with many options and it is essential that it be thoroughly tested to verify that all aspects are functioning correctly. This Section will describe a number of problems which have been analyzed by GEOROC. They have been selected not only to provide assurance that GEOROC is functioning correctly but also to demonstrate typical rock mechanics and engineering applications for which GEOROC may be used. To assist potential users, the input file for each problem is also included.

6.1 CANTILEVER BEAM

To verify that GEOROC is providing correct predictions of mechanical behaviour at the most fundamental level, a simple cantilever beam was analyzed. The problem formulation is shown in Figure 6-1, where a single element simulates a beam with Young's modulus of 690 GPa (10^8 ksi) and Poisson's ratio of 0.20. The beam is fixed at one end and a vertical point load of 89.0 kN (20 kips) is applied at the other end.

Figure 6-2 shows a plot of the vertical deflection of the beam plotted against distance as predicted by GEOROC. Also shown is the analytical solution for this problem (Popov, 1976, p.580). The agreement between the two methods is within 0.4%.

This same problem was re-analyzed but using a number of elements to model the beam. In all these cases GEOROC produced accurate stress and displacement results.

A listing of the input file for the single element analysis is shown in Table 6-1.

6.2 THICK WALLED CYLINDER

Whenever possible, a computer program should be verified by comparison to analytical solutions of problems closely similar to those for which it has been designed. Although analytical solutions to problems as complex as those presented by mine design are not available, the second verification problem

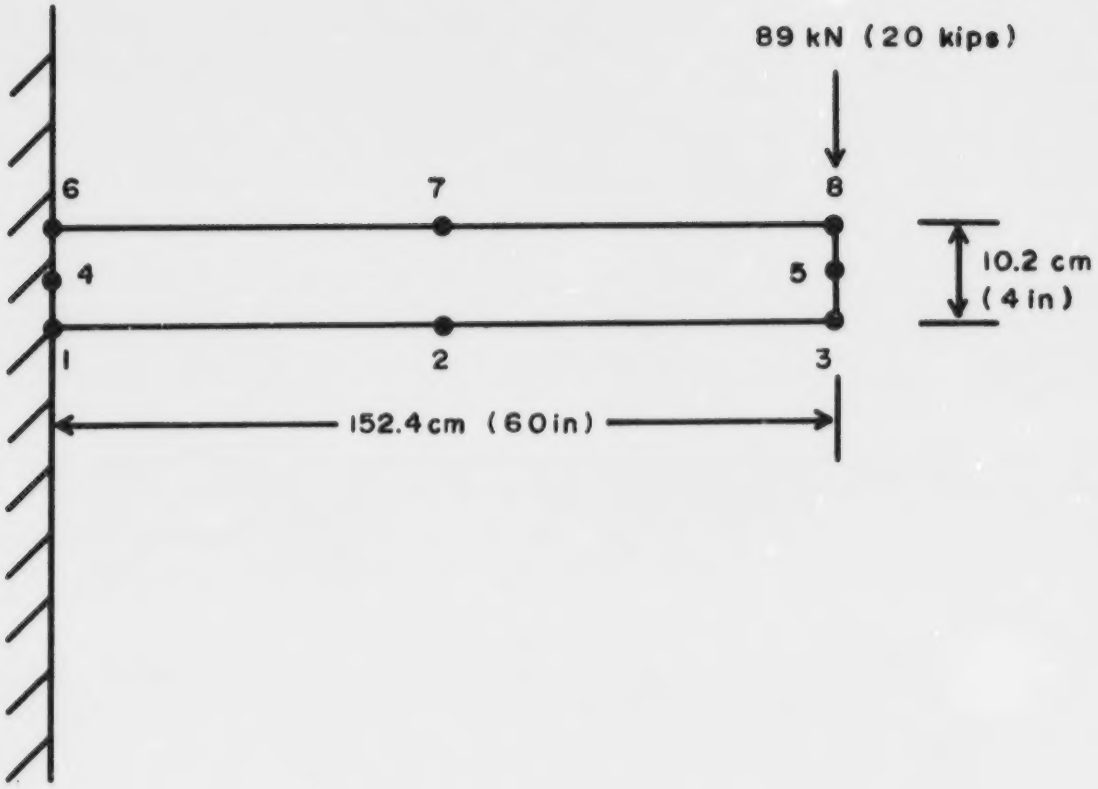


Figure 6-1. The Single Element Cantilever Beam Problem Showing Nodal Positions.

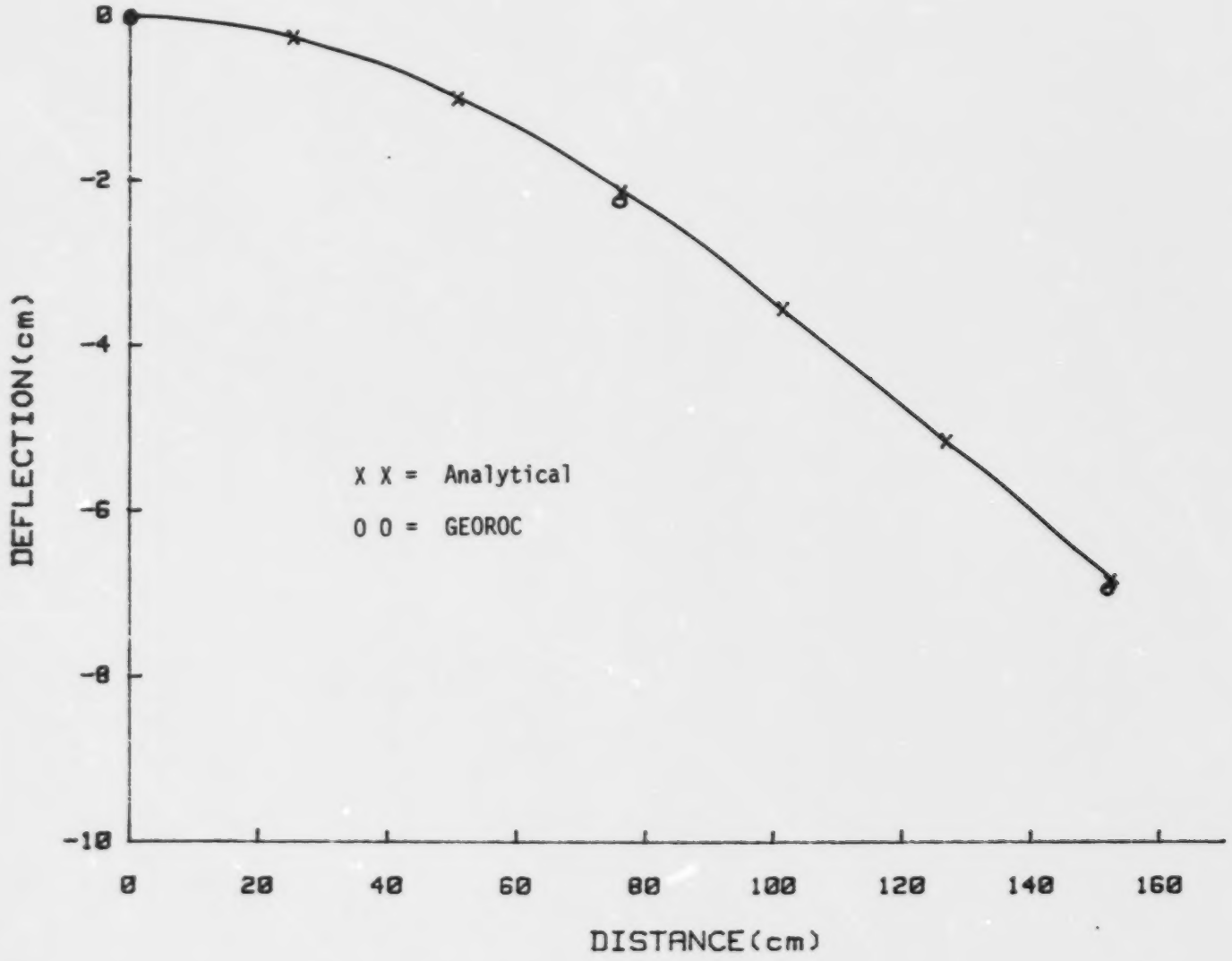


Figure 6-2. Deflection of Cantilever Beam, Shown in Figure 6-1, as a Function of the Distance from the Fixed End.

Table 6-1. Input File for GEOROC for the Cantilever Beam Problem

```
      8     1     2     1     1     2     2     0     0    16
CANTILEVER BEAM EXAMPLE PROBLEM
0
     1     1     1     1     1   123     0     1     0     0     0
     1     0
100000000.     0.2           1.0           0.0
     1     1     1     2     3     5     8     7     6     4
     1          0.            0.
     2         30.            0.
     3         60.            0.
     4          0.            2.
     5         60.            2.
     6          0.            4.
     7         30.            4.
     8         60.            4.
     1    11
     6    11
     1     0     0     0
     8          0.          -20.
     0
     0
```

compares exact and finite element analysis of the viscoelastic behaviour of a continuum under stress. The prediction of stress relaxation and stress-induced time dependent deformation is key to the modelling of potash mine opening.

The problem presented by Greenbaum and Rubinstein (1968) was chosen because both an analytical solution and a finite element approximation are available in the literature and also because of its use of the power law for creep which is considered to be a suitable constitutive law for potash. The problem is a thickwalled, infinitely long tube to which is applied an internal pressure of 2.5 kPa (365 psi); this is illustrated in Figure 6-3.

The material properties used in this model are:

Young's modulus = 137.9 GPa (20×10^6 psi)

Poisson's ratio = .499

$e = 6.4 \times 10^{-18} s^{4.4} t^{0.7}$

where s is effective stress, e is creep strain and t is time.

Analytical solutions to the problem are given for stresses in three dimensions under elastic and steady state creep conditions. GEOROC gives very good agreement with each of these cases. Radial and circumferential stress distributions are shown in Figures 6-4 and 6-5 illustrating the close agreement between the exact solution and the present finite element solution.

Greenbaum and Rubinstein (1968) present finite element solutions for creep deformation caused by internal pressure in the cylinder. These are not compared to an exact solution so are not shown here. However, the present finite element program gives very good agreement with all figures given in the reference.

The input file for this problem is shown in Table 6-2.

6.3 BENCHMARK ANALYSIS

GEOROC is based on RE/SPEC's computer program, SPECTROM-32, which is being used extensively in the U.S. nuclear waste disposal program. As part of that program, SPECTROM-32 has undergone considerable testing as well as comparison to other structural computer programs. In particular, it has

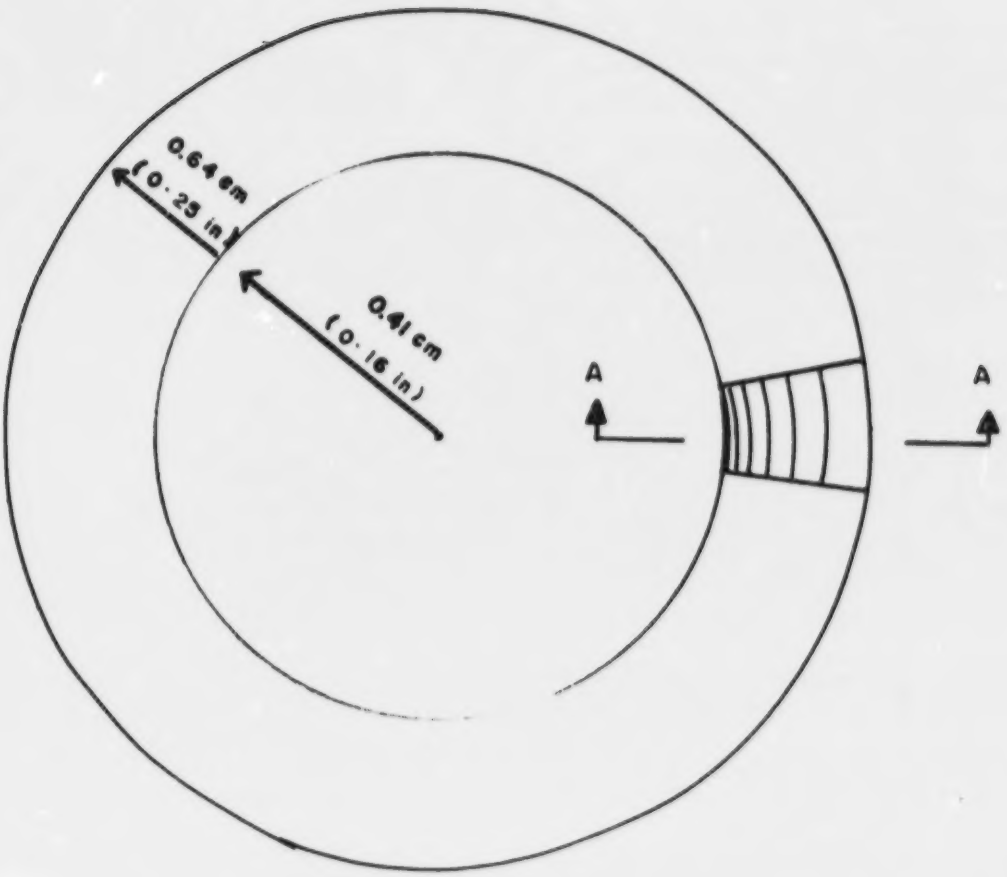


Figure 6-3. Thick Walled Infinite Cylinder : Verification Problem for Finite Element Analysis of Viscoelastic Materials.

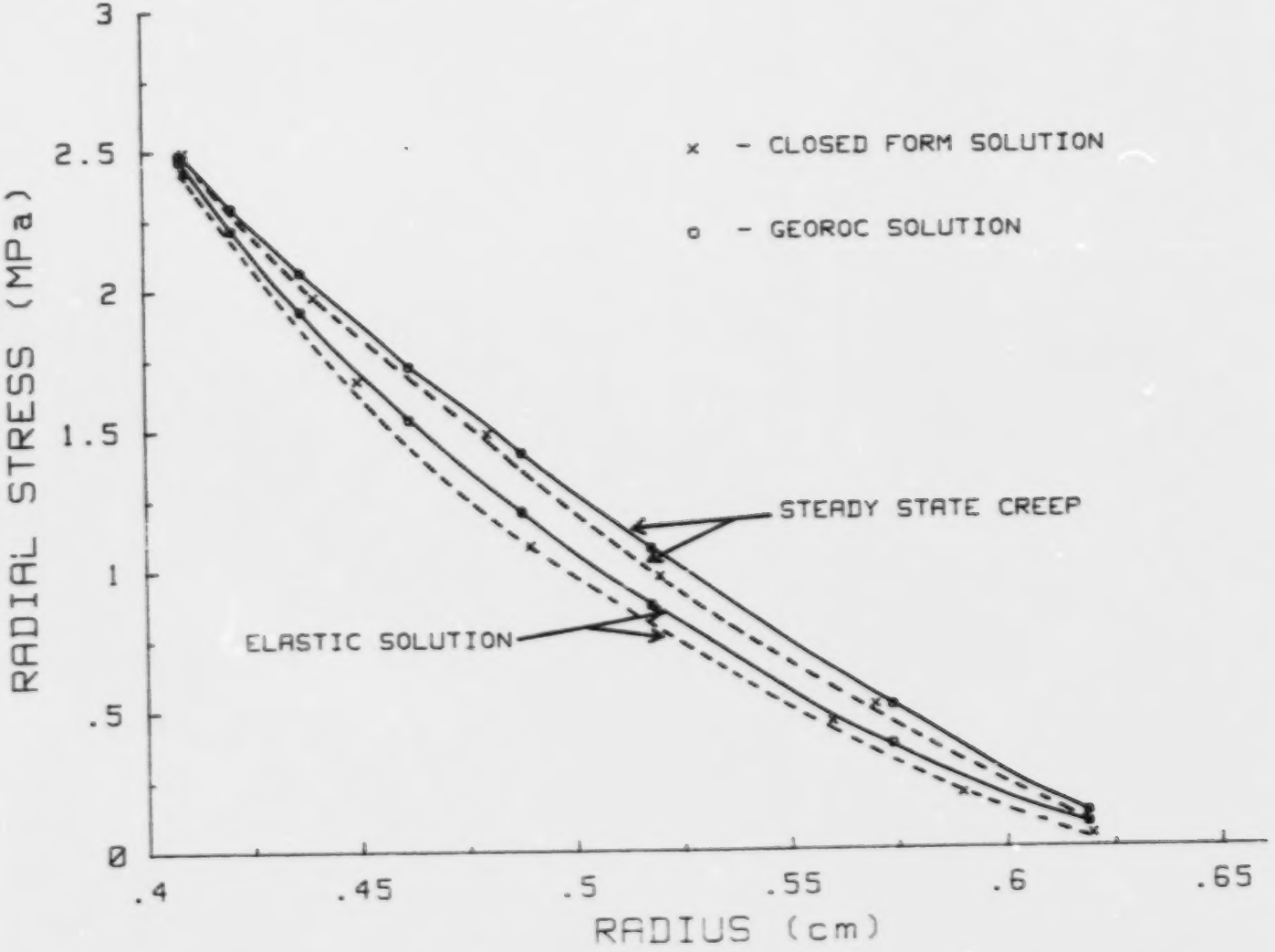


Figure 6-4. Radial Stress as a Function of Radius for the Thick-Walled Cylinder as Predicted by GEOROC and an Analytical Solution

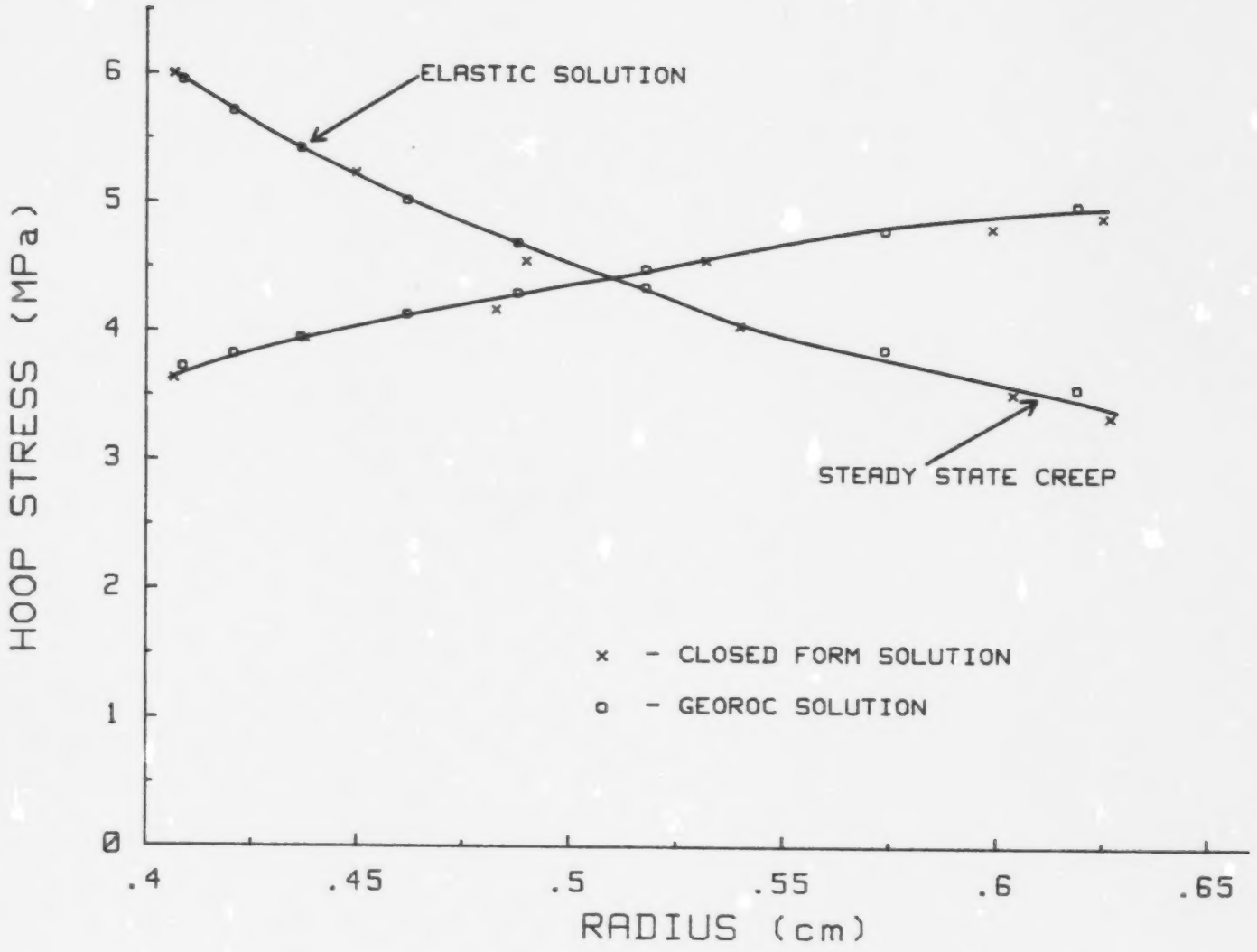


Figure 6-5. Circumferential (Hoop) Stress as a Function of Radius for the Thick-Walled Cylinder as Predicted by GEOROC and an Analytical Solution.

Table 6-2. Input File for GEOROC for the Thick Walled Cylinder Problem

```
   38    7   30    3    1    1    1    0    0   16
    Greenbaum2 Problem Example for GEOROC
0
   50    1
       3.0YEARS
       0.0       0.1
    1    1    1    1   13    2    0   10   38    7    1
       0.1       0.2       0.3       0.4       0.5       1.0       1.5       2.0
       2.5       3.0
    1    2    3    4    5    6    7    8    9   10   11   12   13   14   15   16
   17   18   19   20   21   22   23   24   25   26   27   28   29   30   31   32
   33   34   35   36   37   38
    1    2    3    4    5    6    7
    1    1
20000000.      0.499       0.0       0.0  0.64E-17       4.4       0.70
    1    1    1    2    3    5    8    7    6    4
    2    1    6    7    8   10   13   12   11    9
    3    1   11   12   13   15   18   17   16   14
    4    1   16   17   18   20   23   22   21   19
    5    1   21   22   23   25   28   27   26   24
    6    1   26   27   28   30   33   32   31   29
    7    1   31   32   33   35   38   37   36   34
    1          0.160          0.010
    2          0.160          0.005
    3          0.160          0.000
    4          0.162          0.010
    5          0.162          0.000
    6          0.165          0.010
    7          0.165          0.005
    8          0.165          0.000
    9          0.167          0.010
   10          0.167          0.000
   11          0.170          0.010
   12          0.170          0.005
   13          0.170          0.000
```

Table 6-2. (Continued)

14		0.175	0.010
15		0.175	0.000
16		0.180	0.010
17		0.180	0.005
18		0.180	0.000
19		0.185	0.010
20		0.185	0.000
21		0.190	0.010
22		0.190	0.005
23		0.190	0.000
24		0.195	0.010
25		0.195	0.000
26		0.200	0.010
27		0.200	0.005
28		0.200	0.000
29		0.210	0.010
30		0.210	0.000
31		0.220	0.010
32		0.220	0.005
33		0.220	0.000
34		0.235	0.010
35		0.235	0.000
36		0.250	0.010
37		0.250	0.005
38		0.250	0.000
1	01		
3	01		
4	01		
5	01		
6	01		
8	01		
9	01		
10	01		
11	01		
13	01		
14	01		
15	01		

Table 6-2. (Continued)

```
16    01
18    01
19    01
20    01
21    01
23    01
24    01
25    01
26    01
 8    01
29    01
30    01
31    01
33    01
34    01
35    01
36    01
48    01
 0     1     0     0
 1     1     2     3
   365.0         0.0      365.0         0.0      365.0         0.0
   0
   0
```

been used to analyze the benchmark problems devised by Sandia National Laboratories, which are well documented (Morgan et al., 1981) and have been systematically analyzed by a number of other structural programs. The benchmark analyses by SPECTROM-32 have been reviewed by Sandia Laboratories and are considered to be completely satisfactory. This forms a very stringent and rigorous test since the benchmark problems are very complex and rigorously defined.

6.4 POTASH MINE SIMULATIONS

Two example problems have been devised to illustrate how GEOROC may be used for analyzing rock mechanics problems which are relevant to the Saskatchewan potash industry. These problems are hypothetical, although the room geometries have been chosen to be similar to the instrumentation test site at Cominco Mine.

In the first problem a panel of five rooms is modeled as shown in Figure 6-6. The rooms are located at a depth of 1000 m in the Prairie Evaporite Formation which is assumed to have the same properties for both the potash and salt members. The Dawson Bay dolomite overlies the evaporites. The size of the problem has been reduced by assuming a vertical line of symmetry through the middle of the central room. It is assumed that the initial stress field has the horizontal stress equal to the vertical stress; the latter corresponds to overburden pressure. In this model, room 1 (the outermost) is excavated at time 60 seconds and rooms 2 and 3 are excavated 60 and 90 days after problem initiation. The problem was run to a time of 1 year. Once excavated the room elements were assumed to have the properties of air.

The mesh was generated by GEOEMESH and has 104 elements, 349 nodes and a frontwidth of 64. The input file used to generate this mesh with GEOMESH is shown in Table 6-3. The full mesh is shown in Figure 6-7 and an expanded view of the room region is shown in Figure 6-8.

The inclusion of meaningful rock properties for this analysis was not possible since published values are not available for either the Dawson Bay dolomite or the Prairie Evaporite. The power law was selected for modeling the time dependent evaporite behaviour since the transient creep following

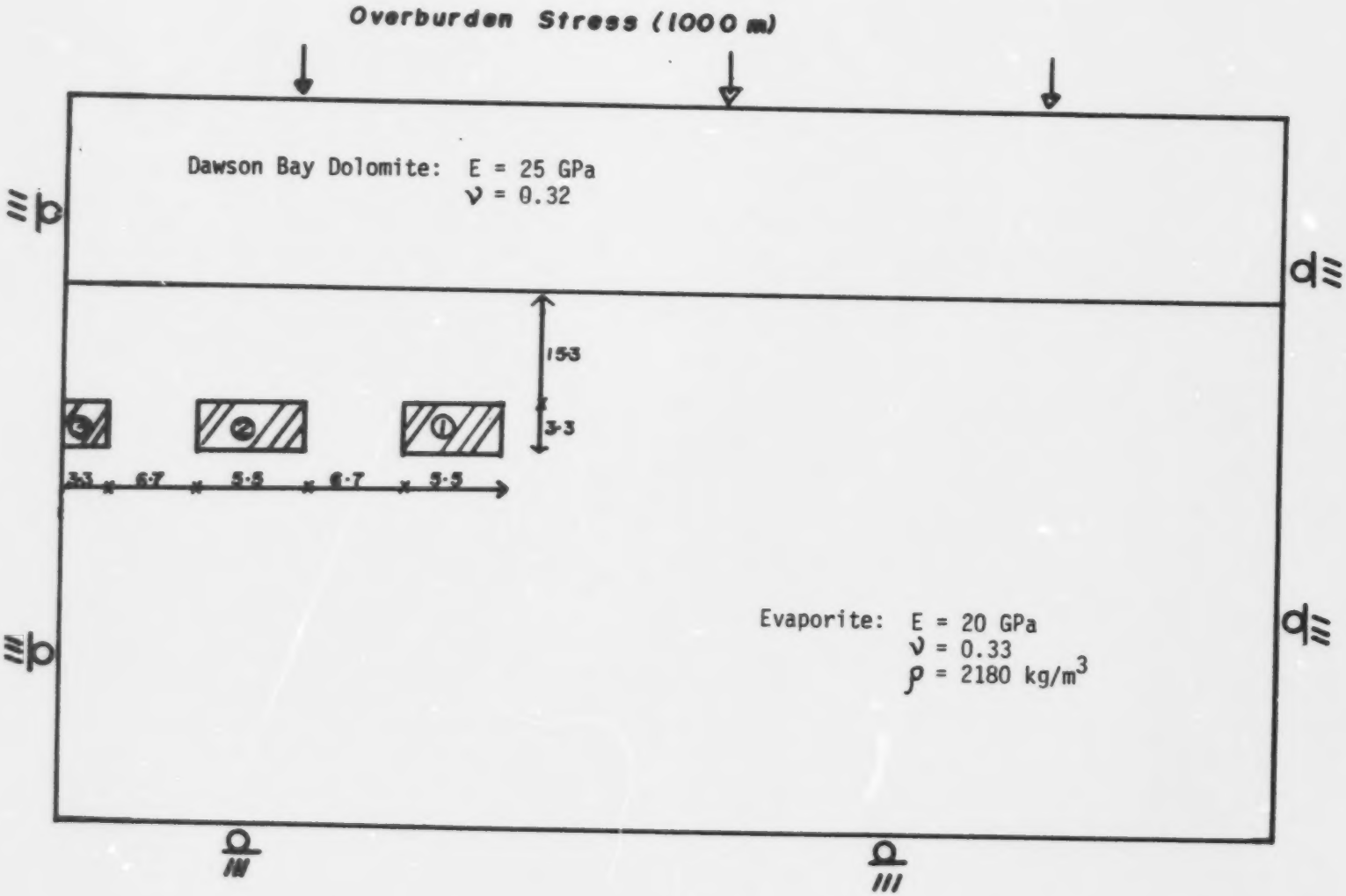


Figure 6-6. Potash Mining Simulation. Based on Cominco Mine.
All Dimensions in Metres.

Table 6-3. Input File for Program GEOMESH to Generate the Mesh shown in Figure 6-7.

```
QMAP,1,  PT1,00.00,16.65, PT2,180.0,16.65, PT3,180.0,60.00, PT4,00.00,60.00, &
 EDGE1,       3.30,16.65,       10.0,16.65,       19.00,16.65,       26.70,16.65, &
             35.00,16.65,       43.3,16.65,       63.30,16.65,       88.00,16.65, &
            127.00,16.65,                                                        &
 EDGE2,     180.00,35.00,                                                        &
 EDGE3,     127.00,60.00,       88.0,60.00,       63.30,60.00,       43.30,60.00, &
             35.00,60.00,       26.7,60.00,       19.00,60.00,       10.00,60.00, &
              3.30,60.00,                                                        &
 EDGE4,       0.00,35.00,                                                        &
 BCM2, 1,                                                                        &
 BCM4, 1,                                                                        &
 MATM, 1

QMAP,2,  PT1,    0,    5, PT2,   31,    5, PT3, 43.3,16.65, PT4,    0,16.65, &
 EDGE1,       3.30, 5.00,       10,    5,       15.50, 5.00,       22.20, 5.00, &
             27.50, 5.00,                                                        &
 NDIV2,  2,                                                                      &
 EDGE3,     35.00,16.65,     26.70,16.65,     19.00,16.65,       10,16.65,       &
              3.30,16.65,                                                        &
 BCM4,1,                                                                         &
 MATM,2,                                                                         &
 EQ3

QMAP,3,  PT1,31.00,-5.00, PT2,43.3,-16.65, PT3,43.30,16.65, PT4,31.00, 5.00, &
 NDIV1, 2,                                                                  &
 EDGE2,     43.30,-7.00,     43.30, 0.00,     43.30, 7.00,                  &
 EDGE4,     31.00, 1.37,        31,    0,     31.00,-1.93,                  &
 EQ3,                                                                       &
 MATM,2

QMAP,4,  PT1,43.3,-16.65, PT2,88.00,-60.0, PT3,88.00,16.65, PT4,43.30,16.65, &
 EDGE1,     63.3,-36.05,                                                    &
 EDGE2,       88,   -31,       88,  -10,       88,    7,                   &
 EDGE3,     63.30,16.65,                                                    &
 EDGE4,     43.30, 7.00,     43.30, 0.00,     43.30,-7.00,                  &
 EQ3, EQ4,                                                                  &
 MATM,2
```

Table 6-3 (Continued). Input File for Program GEOMESH to Generate the Mesh shown in Figure 6-7.

```
QMAP,5,  PT1,   88,  -60, PT2,180.0,-60.0, PT3,180.0,16.65, PT4,88.00,16.65, &
 EDGE1,       127.0,-60.0,                                                    &
 EDGE2,       180.0,-31.0,     180.0,-10.0,     180.0, 7.00,                  &
 EDGE3,       127.0,16.65,                                                    &
 EDGE4,       88.00, 7.00,     88.00,-10.0,     88.00,-31.0,                  &
 EQ3, EQ4,                                                                    &
 BCM1, 0,1,                                                                   &
 BCM2, 1,                                                                     &
 MATM, 2

QMAP,6,  PT1, 0.00,-5.00, PT2,31.00,-5.00, PT3,31.00, 5.00, PT4, 0.00, 5.00, &
 EDGE1,        3.30,-5.00,     10.00,-5.00,     15.50,-5.00,     22.20,-5.00, &
              27.70,-5.00,                                                    &
 EDGE2,       31.00,-1.93,     31.00, 0.00,     31.00, 1.37,                  &
 EDGE3,       27.70, 5.00,     22.20, 5.00,     15.50, 5.00,     10.00, 5.00, &
               3.30, 5.00,                                                    &
 EDGE4,        0.00, 1.37,      0.00, 0.00,      0.00,-1.93,                  &
 EQ2,  EQ3,                                                                   &
 BCM4,1,                                                                      &
 MATM,2

QMAP,7,  PT1, 0.0,-16.65, PT2,43.3,-16.65, PT3,31.00,-5.00, PT4, 0.00,-5.00, &
 NDIV2,2,                                                                    &
 EDGE1,        3.3,-16.65,     10.0,-16.65,     18.5,-16.65,     26.5,-16.65, &
              35.0,-16.65,                                                    &
 EDGE3,       27.70,-5.00,     22.20,-5.00,     15.50,-5.00,     10.00,-5.00, &
               3.30,-5.00,                                                    &
 BCM4,1,                                                                     &
 MATM,2,                                                                     &
 EQ2,  EQ3

QMAP,8,  PT1, 0.00,-60.0, PT2,88.00,-60.0, PT3,43.3,-16.65, PT4, 0.0,-16.65, &
 EDGE1,        3.3,-60,        12.00,-60.0,     30.00,-60.0,     42.0,-60.00, &
                63,-60,                                                      &
 EDGE2,       63.3,-36.05,                                                    &
 EDGE3,       35.0,-16.65,     26.5,-16.65,     18.5,-16.65,     10.0,-16.65, &
               3.3,-16.65,                                                    &
 EDGE4,        0.0,-36.05,                                                    &
 BCM1, 0,1,                                                                   &
 BCM4, 1,                                                                     &
 MATM, 2,                                                                     &
 EQ2, EQ3
 END
```

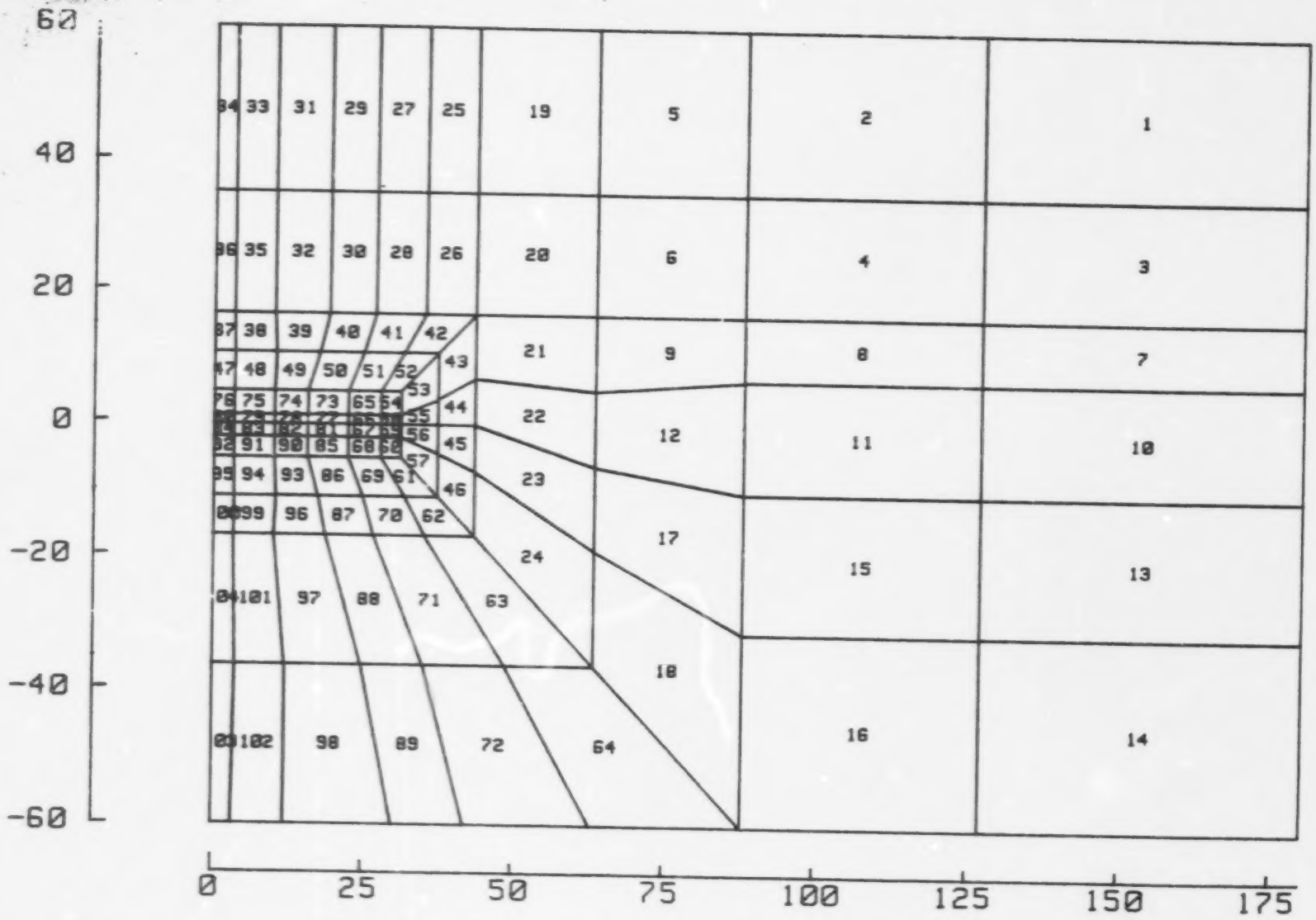


Figure 6-7. Finite Element Mesh for Potash Problem.

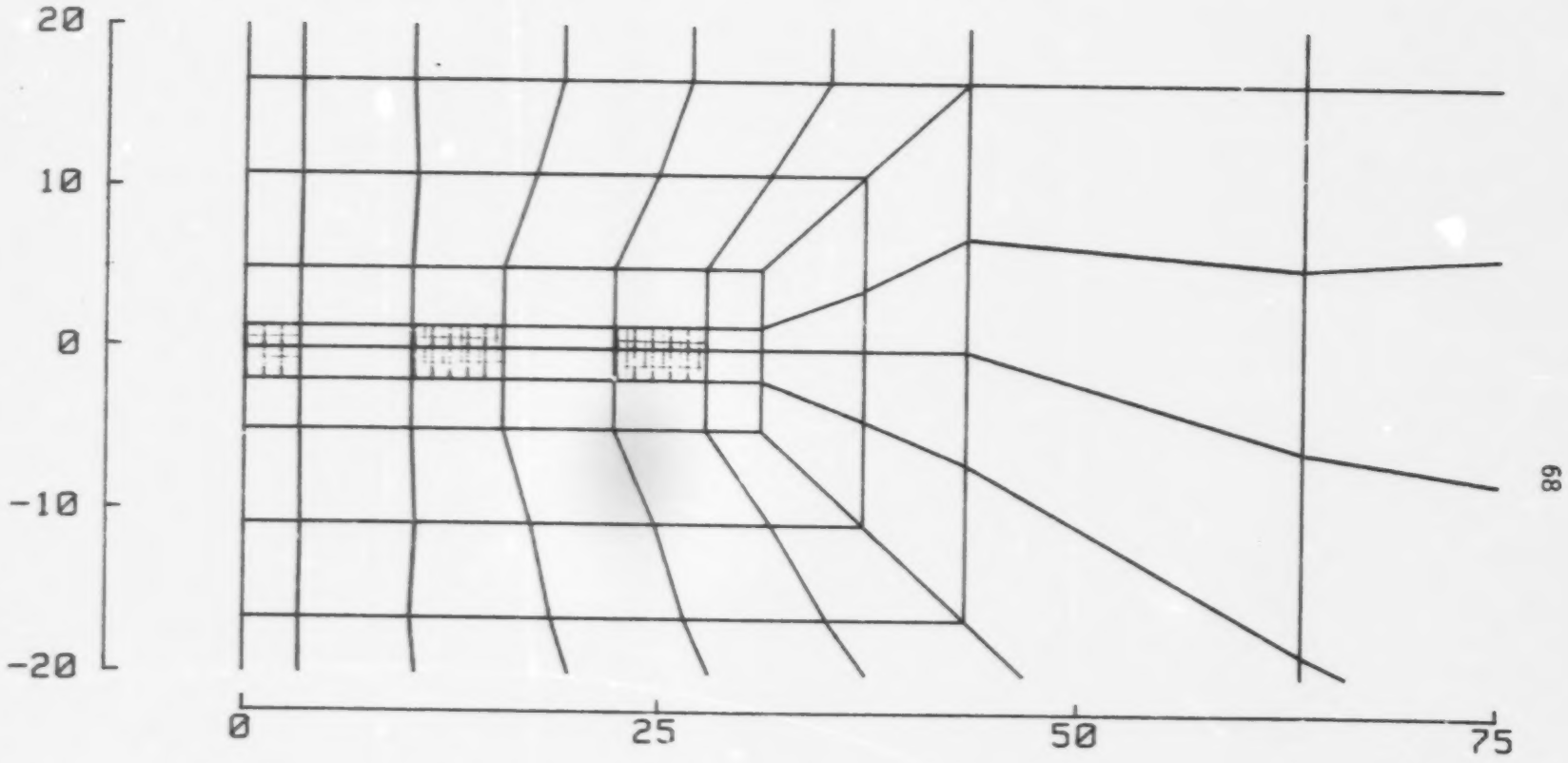


Figure 6-8. Finite Element Mesh for Potash Problem Showing Blow-up of Central Portion of Figure 6-7.

excavation would be included. The parameters for the power law were chosen, rather arbitrarily, to be similar to those measured for the Lotsberg Formation in Alberta. Elastic properties for the evaporite and dolomite were selected arbitrarily from the literature. The dolomite modulus was reduced somewhat from the literature values which are based on laboratory testing to make it more representative of the field modulus. All rock properties are listed in Table 6-4. It is emphasized that rock properties vary dramatically and, thus, the results of this analysis cannot be expected to accurately represent the behaviour at Cominco Mine, although the trends should be correct.

Due to the large quantity of output generated by GEOROC, no reproductions of actual output will be shown. However, pictorial summaries of the output data are presented. Figure 6-9 shows the vertical closure of the three rooms measured at the centre line as a function of time. No backfill was emplaced for this study. It is seen that the two central rooms (2 and 3) undergo substantially less vertical closure than, and are 'protected' by, the outermost pair of rooms. Although the amplitudes of the predicted closures are not expected to agree with the observed closures, the trends should be correct.

Figure 6-10 shows the impact that backfilling with crushed tailings would have. Room 1 was backfilled completely with crushed tailings having the properties shown in Table 6-4 at time 130 days (70 days after mining) and room 2 was backfilled at time 180 days (90 days after mining). The backfill proeprties were determined by laboratory testing (Beddoes, 1984). This analysis simulates backfill that would actually be emplaced immediately after mining rooms 1 and 2. It was assumed that it would take a period of 70 to 90 days for creep closure to consolidate the tailings and yield the properties given in Table 6-3. It is seen that the backfill effectively stops the closure with time in rooms 1 and 2 and decreases the total closure of room 3 by about 20%, even though it was not backfilled.

The effective stress (that which causes creep) is plotted as a function of distance in Figure 6-11 at 70 and 365 days. The migration of effective stress away from the mined rooms into the barrier pillar with time is also shown. It is seen that by 365 days the effective stress extends considerably

Table 6-4. Rock Properties Used in the Potash Mine Simulation

	Dawson Bay Fm.	Evaporite	Backfill
Young's Modulus (GPA)	25	20	0.875
Poisson's Ratio	0.32	0.33	0.4
Density (Kg/m^3)	2500	2180	1920
Leading Coefficient	-	2.605×10^{-11}	-
Power on Stress	-	7.5	-
Power on Time	-	0.39	-

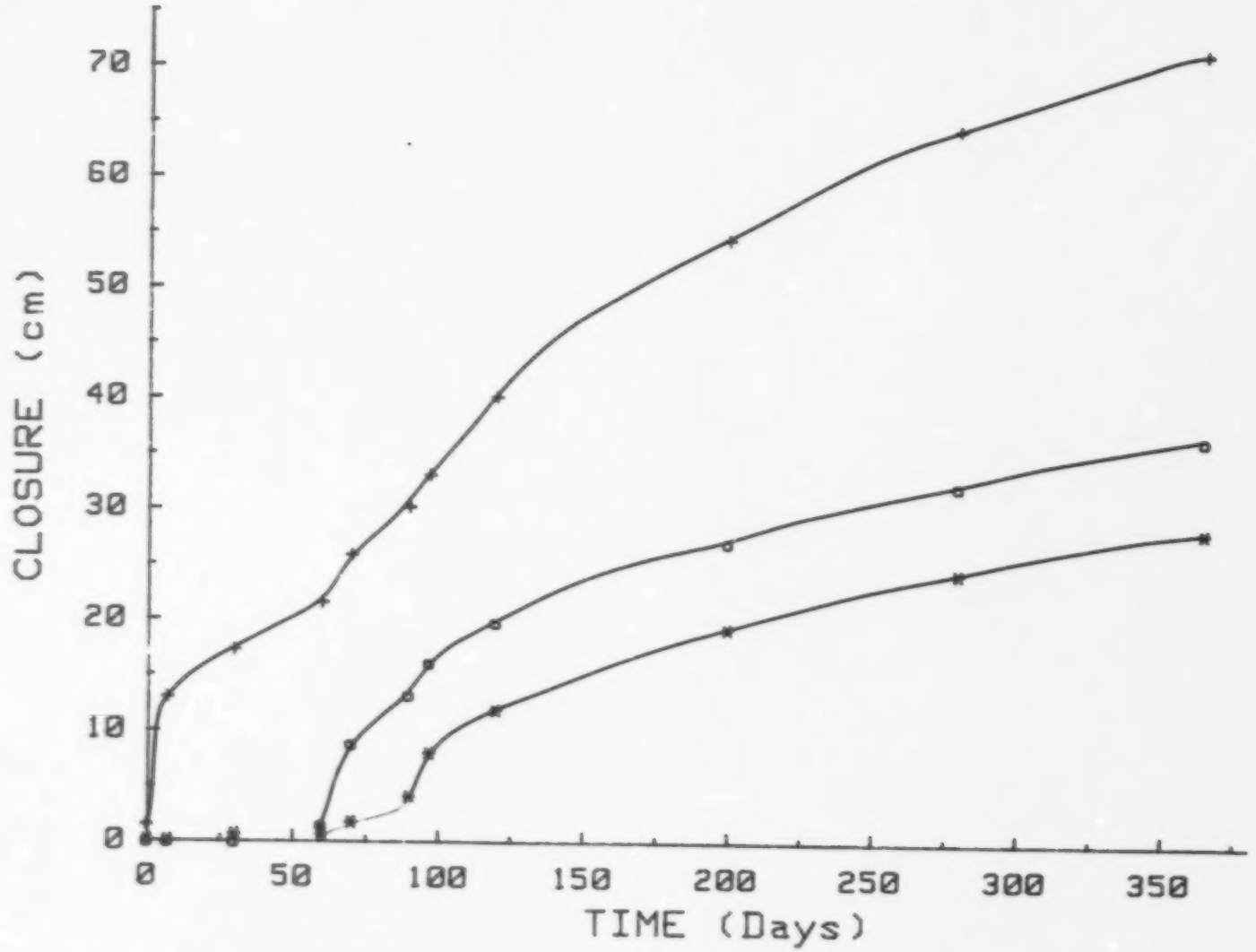


Figure 6-9. Vertical Creep Closure of Rooms One(+), Two(o), and Three(*) as a Function of Time. No Backfill.

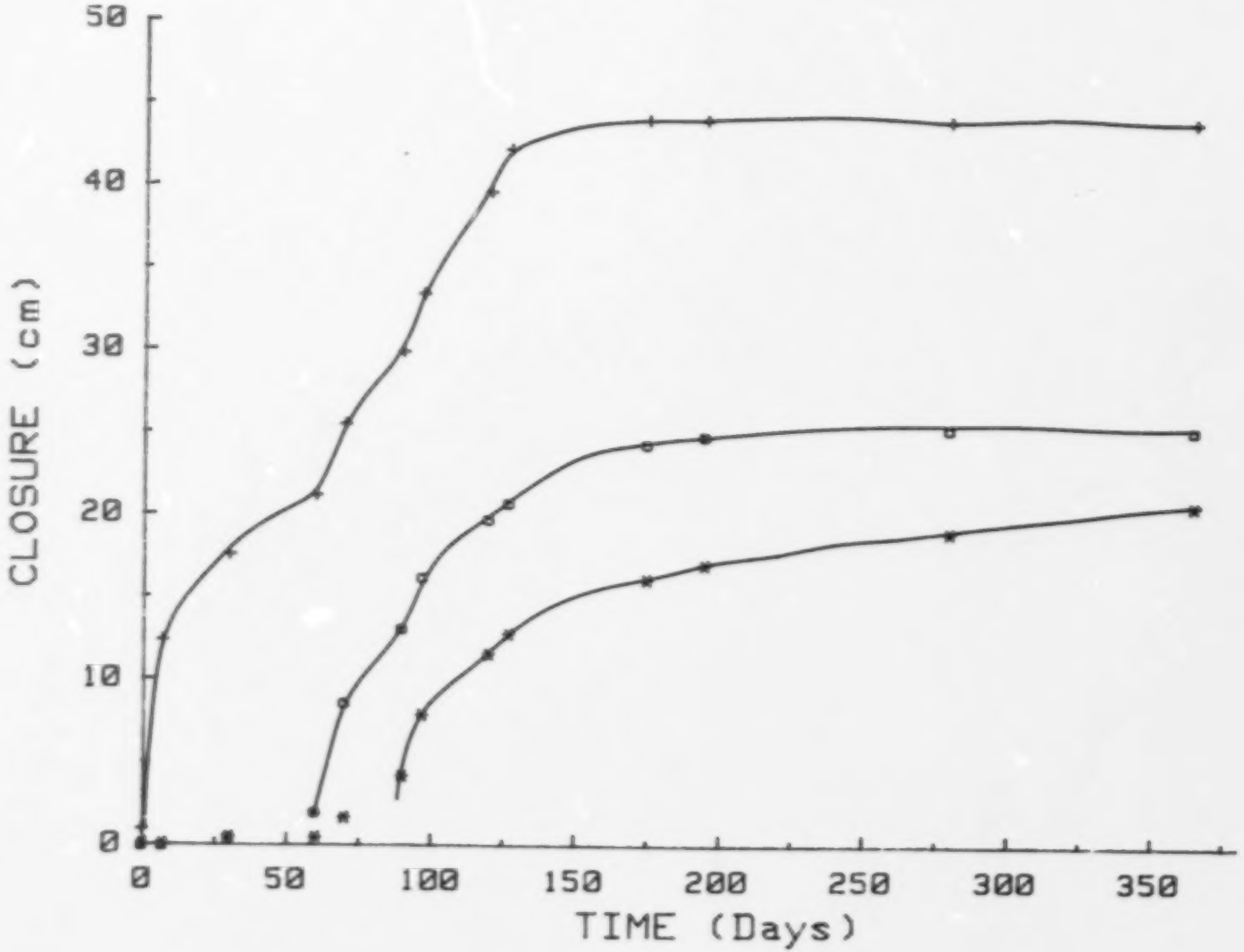


Figure 6-10. Vertical Closure of Rooms One(+), Two(o) and Three(*) as a Function of Time. Room One was Backfilled at 130 Days and Room Two at 180 Days.

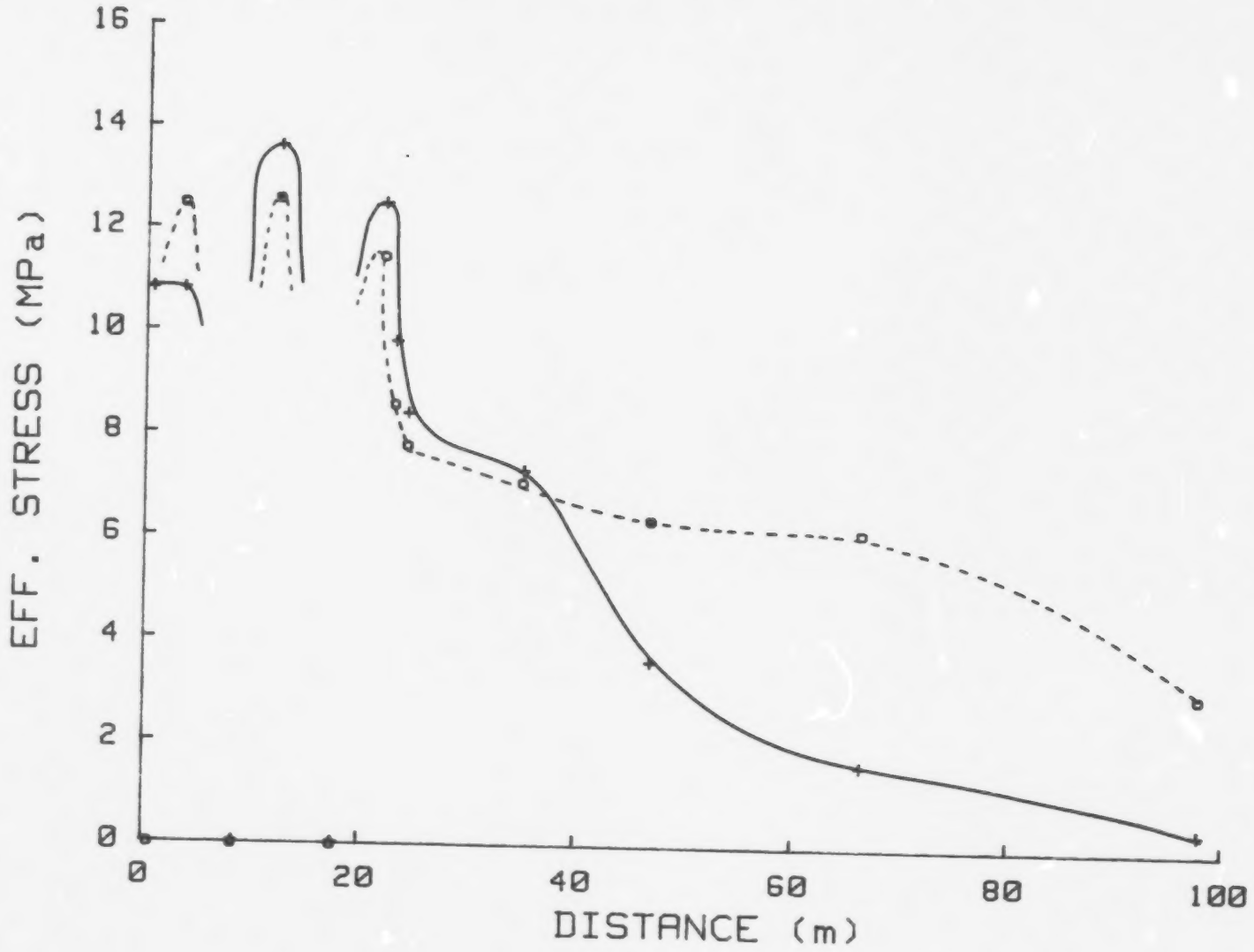


Figure 6-11. Effective Stress as a Function of Horizontal Distance. The Profile is at Mining Level and Starts at the Middle of Room Three. The Points ,+, are for Time 70 Days and Points ,o, are for 365 Days.

further into the rock mass than at 70 days. The input files for GEOROC used for these analyses are listed in Tables 6-5 and 6-6. To save space, not all the mesh (noded and elemental) data are shown.

The user-created files needed to run the GNP post processors to create vertical room closure versus time plots similar to Figure 6-9 are shown in Table 6-7. The steps to implement the procedure are outlined in Table 6-8. Note that the files in Table 6-7 also create horizontal room closure plots which are not shown in this report (to conserve space). Also note that the files in Table 6-7 create printer plots. To understand the steps outlined in Table 6-8, it will be helpful to also refer to Figure 5-1.

In the VAX VMS operating system, command files can be used to automatically implement many of the steps shown in Table 6-8, and also to provide unique file names.

Table 6-5. Input File for GEOROC for the Potash Problem (No Backfill)

```
349  104   53    2    3    1    1    0    0   64
 FIVE ROOM POTASH CASE, WITHOUT ANY BACKFILL AT ALL.
  0
5000    1
     365. DAYS
      0.0          1.
  1    0    0    0   13    2    0   10   48   28    1
       7.        30.        60.        70.        90.        97.       120.       200.
     280.       365.
252  253  256  295  296  294  266  268  249  248  246  292  290  288  263  260
244  207  206  271  214  212  258  209  117  120  123  126  129  135   25   19
307  306  302  308  297  241  240  149  141  119  156  233  232  158  230  229
 76   92   79   83   74   90   77   81   65   68   58   59   56   45   12   10
 39   32   31   93   97   98   66   67   78   82   80   84
  1    0
  25000.       0.32        0.0       .0025
  2    1
  20000.       0.33        0.0       .00218 2.605E-11        7.5        0.55
  3    0
    835.       0.499       0.0       .00192
  1    1   10    8    7    2    3    1    5    4    0
  2    1   16   11   10    4    5    6   13   14    0
  3    1   22   20   19    9    7    8   10   12    0
  4    1   25   23   22   12   10   11   16   17    0
  5    1   62   18   16   14   13   15   59   60    0

                *** Lines 6-100 Omitted ***

101    2  347  336  332  329  330  331  339  337    0
102    2  342  334  333  335  332  336  347  349    0
103    2  343  344  342  349  347  346  348  345    0
104    2  348  346  347  337  339  338  341  340    0
  1      153.500000        60.000000
  2      180.000000        47.500000
  3      180.000000        60.000000
  4      127.000000        47.500000
  5      127.000000        60.000000
  6      107.500000        60.000000
  7      180.000000        35.000000
```

Table 6-5. (Continued)

```
  8      153.500000        35.000000
  9      180.000000        25.825000
 10      127.000000        35.000000
 11      107.500000        35.000000
 12      127.000000        25.925000
 13       88.000000        60.000000

        *** Lines 14-339 Omitted ***

340        0.000000       -26.350000
341        0.000000       -16.650000
342        3.300000       -60.000000
343        0.000000       -60.000000
344        1.650000       -60.000000
345        0.000000       -48.025000
346        1.863925       -36.049939
347        3.727845       -36.049878
348        0.000000       -36.050000
349        3.645972       -48.024939
  2   10         0.000000          0.000000
  3   10         0.000000          0.000000
  7   10         0.000000          0.000000
  9   10         0.000000          0.000000
 19   10         0.000000          0.000000
 21   10         0.000000          0.000000
 28   10         0.000000          0.000000
 30   10         0.000000          0.000000
 37   10         0.000000          0.000000
 39   10         0.000000          0.000000

        *** Lines 40-343 Omitted ***

344   01         0.000000          0.000000
345   10         0.000000          0.000000
348   10         0.000000          0.000000
```

Table 6-5. (Continued)

```
 9     0     0     1
      0.0          9.81
 1        -22.1
      60.          -60.      .0023      0.99      1.01
 3
 2     6          0.0007
66    67
68   212   214   271
      0.0          0.0       0.0       0.0       0.0       0.0
59   209   211   212
      0.0          0.0       0.0       0.0       0.0       0.0
58   206   208   209
      0.0          0.0       0.0       0.0       0.0       0.0
65   244   207   206
      0.0          0.0       0.0       0.0       0.0       0.0
77   258   243   244
      0.0          0.0       0.0       0.0       0.0       0.0
81   271   259   258
      0.0          0.0       0.0       0.0       0.0       0.0
 2     6          60.
78    82
90   288   290   292
      0.0          0.0       0.0       0.0       0.0       0.0
81   260   261   288
      0.0          0.0       0.0       0.0       0.0       0.0
77   246   247   260
      0.0          0.0       0.0       0.0       0.0       0.0
74   249   248   246
      0.0          0.0       0.0       0.0       0.0       0.0
79   263   250   249
      0.0          0.0       0.0       0.0       0.0       0.0
83   292   264   263
      0.0          0.0       0.0       0.0       0.0       0.0
 2     4          90.
80    84
92   294   296   295
      0.0          0.0       0.0       0.0       0.0       0.0
83   268   270   294
      0.0          0.0       0.0       0.0       0.0       0.0
79   256   254   268
      0.0          0.0       0.0       0.0       0.0       0.0
76   252   253   256
      0.0          0.0       0.0       0.0       0.0       0.0
```

Table 6-6. Input File for GEOROC for the Potash Problem (with Backfill)

```
349  104   53    2    3    1    1    0    4   64
 FIVE ROOM POTASH CASE, WITH BACKFILL
  0
5000    1
    365. DAYS
     0.0          1.
  1    0    0    0   13    2    0   10   48   28    1
       7.        30.        60.        70.        90.        97.       120.       200.
     280.       365.
252  253  256  295  296  294  266  268  249  248  246  292  290  288  263  260
244  207  206  271  214  212  258  209  117  120  123  126  129  135   25   19
307  306  302  308  297  241  240  149  141  119  156  233  232  158  230  229
 76   92   79   83   74   90   77   81   65   68   58   59   56   45   12   10
 39   32   31   93   97   98   66   67   78   82   80   84
  1    0
  25000.       0.32         0.0        .0025
  2    1
  20000.       0.33         0.0        .00218 2.605E-11        7.5        0.55
  3    0
    875.       0.4          0.0        .00192
  1    1   10    8    7    2    3    1    5    4    0
  2    1   16   11   10    4    5    6   13   14    0
  3    1   22   20   19    9    7    8   10   12    0
  4    1   25   23   22   12   10   11   16   17    0

                    *** Lines 5-99 Omitted ***

100    2  341  338  339  328  325  326  324  327    0
101    2  347  336  332  329  330  331  339  337    0
102    2  342  334  333  335  332  336  347  349    0
103    2  343  344  342  349  347  346  348  345    0
104    2  348  346  347  337  339  338  341  340    0
  1      153.500000        60.000000
  2      180.000000        47.500000
  3      180.000000        60.000000
  4      127.000000        47.500000
  5      127.000000        60.000000
  6      107.500000        60.000000
```

Table 6-6. (Continued)

```
  7      180.000000        35.000000
  8      153.500000        35.000000
  9      180.000000        25.825000
 10      127.000000        35.000000
 11      107.500000        35.000000
 12      127.000000        25.825000
 13       88.000000        60.000000

      *** Lines 14-339 Omitted ***

340        0.000000       -26.350000
341        0.000000       -16.650000
342        3.300000       -60.000000
343        0.000000       -60.000000
344        1.650000       -60.000000
345        0.000000       -48.025000
346        1.863925       -36.049939
347        3.727845       -36.049878
348        0.000000       -36.050000
349        3.645972       -48.024939
  2   10        0.000000         0.000000
  3   10        0.000000         0.000000
  7   10        0.000000         0.000000
  9   10        0.000000         0.000000
 19   10        0.000000         0.000000
 21   10        0.000000         0.000000
 28   10        0.000000         0.000000
 30   10        0.000000         0.000000
 37   10        0.000000         0.000000
 39   10        0.000000         0.000000

      *** Lines 40-342 Omitted ***

343   11        0.000000         0.000000
344   01        0.000000         0.000000
345   10        0.000000         0.000000
348   10        0.000000         0.000000
  0    0    0    1
     0.0       9.81
```

Table 6-6. (Continued)

```
:      -22.1
      60.          -60.        .0023        0.99         1.01
 3
 2     6          0.0007
66    67
68   212   214   271
       0.0          0.0         0.0          0.0          0.0          0.0
59   209   211   212
       0.0          0.0         0.0          0.0          0.0          0.0
58   206   208   209
       0.0          0.0         0.0          0.0          0.0          0.0
65   244   207   206
       0.0          0.0         0.0          0.0          0.0          0.0
77   258   243   244
       0.0          0.0         0.0          0.0          0.0          0.0
81   271   259   258
       0.0          0.0         0.0          0.0          0.0          0.0
 2     6           60.
78    82
90   288   290   292
       0.0          0.0         0.0          0.0          0.0          0.0
81   260   261   288
       0.0          0.0         0.0          0.0          0.0          0.0
77   246   247   260
       0.0          0.0         0.0          0.0          0.0          0.0
74   249   248   246
       0.0          0.0         0.0          0.0          0.0          0.0
79   263   250   249
       0.0          0.0         0.0          0.0          0.0          0.0
83   292   264   263
       0.0          0.0         0.0          0.0          0.0          0.0
 2     4           90.
80    84
92   294   296   295
       0.0          0.0         0.0          0.0          0.0          0.0
83   268   270   294
       0.0          0.0         0.0          0.0          0.0          0.0
79   256   254   268
       0.0          0.0         0.0          0.0          0.0          0.0
76   252   253   256
       0.0          0.0         0.0          0.0          0.0          0.0
       0.0
66     3          130.
67     3          130.
78     3          180.
```

Table 6-7. Files Required to run GNP for Plots Similar to Figure 6-9. (a) shows Input for ALGEBRA and (b) shows Input for TPLOT.

(a)

```
HTCLOSR1 = DISPLY$214 - DISPLY$207
HTCLOSR2 = DISPLY$290 - DISPLY$248
HTCLOSR3 = DISPLY$295 - DISPLY$252
WDTHCLOSR1 = DISPLX$258 - DISPLX$209
WDTHCLOSR2 = DISPLX$263 - DISPLX$260
WDTHCLOSR3 = 2.0 * (-DISPLX$268)
END
```

(b)

```
HTCLOSR1
HTCLOSR2
HTCLOSR3
WDTHCLOSR1
WDTHCLOSR2
WDTHCLOSR3
PRINTER
END
```

Table 6-8. Steps Required to Create Plots Similar to Figure 6-9 Using GNP

1. Create file shown in Table 6-7(a) and name it ALGEBRA_INPUT.DAT

2. Rename TAPE22 (GEOROC output) to ALGEBRA_BIN_INPUT.DAT

3. Run ALGEBRA. It produces a file named ALGEBRA_BIN_OUTPUT

4. Rename ALGEBRA_BIN_OUTPUT to TPLOT_BIN_INPUT.DAT

5. Create file shown in Table 6-7(b) and name it TPLOT_INPUT.DAT

6. Run TPLOT. It will generate 6 printer plots showing vertical and horizontal room closure versus time.

7 CONCLUSIONS/RECOMMENDATIONS

Although GEOROC is a powerful analysis tool in its present form, there are a number of areas in which it can be further improved. A major objective in modern programming technique is to make software very easy to use i.e., user-friendly. The preparation of GEOROC's input file would be greatly simplified if a small interactive program were developed which queries the user for the input data, checks for errors and then builds the input file. This would also help eliminate typographical and careless errors which at present are relatively easy to make and often difficult to detect.

Time-dependent problems of duration 6 months or longer, which are common in potash mining, require considerable computational time. For example, the potash problems described in Section 6.4 required approximately 2-3 hours of CPU time on a VAX 11/750 computer. Clearly, an important objective in the future evolution of GEOROC is to improve the numerical analysis methods so that computational efficiency can be improved. At present RE/SPEC is investigating a four-noded element with pressure-correction that will have the capability to accurately model viscoelastic behaviour and may reduce computational time by about 75%.

The technical capabilities of GEOROC could also be extended in the future. For example it would be useful to add the capability to model slip lines (e.g. clay seams, joints), the plastic behaviour of rock and stress distribution after failure occurs, and thermal stresses. These capabilities are generally available and, if incorporated into GEOROC, would greatly enhance its usefulness in other underground applications such as hard rock mining, coal mining, nuclear waste disposal studies, and so on. It is important that GEOROC continue to evolve and improve its capabilities with time.

GEOROC is a versatile and powerful tool which has a wide range of potential applications in potash rock mechanics and mine design. Many areas which previously could not be readily studied such as total extraction mining, the mining of multiple potash seams, the use of backfill for support, potential brittle failure in overlying strata, to name only a

few, can now be addressed.

However, it must be realized that the final results are directly related to the quality of the input data used. At present, very little is known of the mechanical properties of the Prairie Evaporite salt and potash units, or of the overlying Dawson Bay Formation. It is urged that an integrated study be undertaken where a carefully monitored underground excavation be analyzed by GEOROC. In parallel, a detailed laboratory investigation of the rock proeprties should be undertaken. Particular emphasis should be placed on determining the time dependent constitutive laws for salt and potash. This kind of study is essential for defining the necessary input parameters for GEOROC, for becoming familiar with and fine-tuning GEOROC's ability to model relevant potash situations, and, ultimately, for providing confidence that GEOROC will provide accurate reliable predictive capability for the Saskatchewan potash mining community. By incorporating some of the capabilities discussed above, GEOROC can also become a powerful tool for a whole range of underground applications.

8 REFERENCES

Beddoes, R.J., 1984, Determination of Engineering Properties of Waste Salt for Backfilling in Underground Potash Mines, Prepared by RE/SPEC Ltd., Calgary, AB., RSL-004 for Canada Centre for Mineral and Energy Technology, Energy, Mines and Resources, Canada.

Carter, N.L., 1981, Creep and Creep-Rupture of Rocksalt, ONWI-224, Prepared by RE/SPEC Inc., Rapid City, S.D., RSI-0139, for Office of Nuclear Waste Isolation, Battelle Memorial Institute, Columbus, OH.

Desai, C.S. and J.F. Abel, 1972, Introduction to the Finite Element Method, Van Nostrand Reinhold, New York.

Gangi, A.F., 1981, A Constitutive Equation for One-Dimensional Transient and Steady-State Flow of Solids, in The Mechanical Behaviour of Crustal Rocks, AGU Monograph No. 24, Washington, D.C.

Greenbaum, G.A. and M.F. Rubinstein, 1968, Creep Analysis of Axisymmetric Bodies Using Finite Elements, Nuclear Engineering Design, Vol. 7, pp. 379-397.

Hansen, F.D., 1976, Experimental Consolidation of Granulated Rock Salt with Application to Sleeve Buckling, Prepared by RE/SPEC Inc., Rapid City, SD, RSI-0044, for the Oakridge National Laboratory under Subcontract No. 4269 with Union Carbide Corporation, Nuclear Division, ORNL-SUB-4269-21, Oak Ridge, TN.

Hansen, F.D., and K.D. Mellegard, 1979, Creep Behaviour of Bedded Salt From Southeastern New Mexico at Elevated Temperatures, SAND79-7030, Prepared by RE/SPEC Inc., Rapid City, SD, RSI-0062, for Sandia National Laboratories, Albuquerque, NM.

Hardy, H.R. Jr., 1982, Theoretical and Laboratory Studies Relative to the Design of Salt Caverns for the Storage of Natural Gas, American Gas Association, Arlington, VA.

Krieg, R.D., 1980, A Unified Creep-Plasticity Model for Halite, SAND80-1195, Sandia National Laboratories, Albuquerque, NM.

Mellegard, K.D., P.E. Senseny and F.D. Hansen, 1983, Quasi-Static Strength and Creep Characteristics of 100-MM-Diameter Specimens of Salt from Avery Island, Louisiana. Prepared by RE/SPEC Inc., Rapid City, SD, RSI-0140, for Office of Nuclear Waste Isolation under Subcontract No. E512-02300 with Battelle Memorial Institute, Columbus, OH.

Morgan, H.S., R. D. Krieg and R.Y. Matalucci, 1981, Comparative Analysis of Nine Structural Codes Used in the Second WIPP Benchmark Problem, SAND81-1389, Sandia National Laboratories, Albuquerque, NM.

Munson, D.E., and P.R.R Dawson, 1979, Constitutive Model for the Low-Temperature Creep of Salt (with Application to WIPP), SAND79-1853, Sandia National Laboratories, Albuquerque, NM.

NCAR (National Center for Atmospheric Research), 1983, Technical Notes, NCAR/TN-161-1A through NCAR/TN-166-1A, Boulder, CO.

Owen, D.R.J. and E. Hinton, 1980, Finite Elements in Plasticity: Theory and Practice, Pineridge Press, Swansea.

Pfeifle, T.W., K.D. Mellegard and P.E. Senseny, 1982, Preliminary Constitutive Properties for Salt and Nonsalt Rocks from Four Potential Repository Sites, ONWI-450, Prepared by RE/SPEC Inc., Rapid City, SD, RSI-0191, for Battelle Memorial Institute, Columbus, OH.

Popov, E.V., 1976, Mechanics of Materials, Prentice-Hall, New York.

Preece, D.S. and W.R. Wawersik, 1984, Leached Cavern Design Using a Fracture Criterion for Rock Salt, in Proceedings Twenty-Fifth U.S. Rock Mechanics Symposium, Northwestern University, IL, Society of Mining Engineers, New York.

Sego, D.C. and N.R. Morgenstern, 1983, Deformation of Ice Under Low Stresses, Canadian Geotechnical Journal, Vol. 20, pp. 587-602.

Senseny, P.E., 1983, Review of Constitutive Laws Used to Describe the Creep of Salt, ONWI-295, Prepared by RE/SPEC Inc., Rapid City, SD, RSI-0151, for the Office of Nuclear Waste Isolation under Subcontract No. E512-02300 with Battelle Memorial Institute, Columbus, OH.

Van Sambeek, L.L., 1984, Potential Reduction in Surface Subsidence by Backfilling or Flooding the Belle Isle Mine, Prepared by RE/SPEC Inc., Rapid City, SD, RSI-0247.

Wawersik, W.R. and D.W. Hannum, 1979, Interim Summary of Sandia Creep Experiments on Rock Salt from the WIPP Study Area, Southeastern New Mexico, SAND79-0115, Sandia National Laboratories, Albuquerque, NM.

Wawersik, W.R. and D.W. Hannum, 1980, Mechanical Behaviour of New Mexico Rock Salt in Triaxial Compression Up to 200°C, Journal of Geophysical Research, Vol. 85, pp. 891-900.

Webster, G.A., A.P.D. Cox and J.E. Dorn, 1969, A Relationship Between Transient and Steady-State Creep at Elevated Temperatures, Metal Sci.J., Vol. 3, pp. 221-225.

Zienkiewicz, O.C., 1977, The Finite Element Method in Engineering Science, McGraw-Hill, New York.

9 ACKNOWLEDGEMENTS

Program GEOROC was based to a large extent on program SPECTROM-32 and its predecessor's developed by RE/SPEC Inc., Rapid City, SD. The authors are grateful to the many individuals at RE/SPEC who have contributed over the years to the evolution of these programs. Mr. Y.S. Yu of CANMET was the Scientific Authority of this project and his many discussions have helped improve the final report. Mr. R.J. Beddoes, RE/SPEC Ltd., has contributed to a number of aspects in this study. Thanks are expressed to the Coal Research Laboratory, CANMET, Calgary for providing time on their VAX11/750 computer and assistance with the system. Finally, we would like to express our gratitude to CANMET for funding this project through the Short Term Assistance in Research and Technology (START) program and for the support of the potash industry in general.

APPENDIX A

AN ALPHABETICAL LIST BRIEFLY DESCRIBING ALL GEOROC SUBROUTINES AND FUNCTIONS

A.1 OVERALL PROGRAM STRUCTURE AND CONTROL

MAIN — The main program allows the user to define the storage space for 'ARRAY', the storage area for the majority of arrays in GEOROC and then calls the main subroutine GEOROC to run the program.

GEOROC — This is the real 'main program', which calls PARAMETERS to input the first card, SIZE to map all of the arrays into ARRAY, NULL to set arrays to zero, INDATA to input and echo the rest of the input, EMAT to initialize elasticity matrices for each material type, and finally STRUCPROC to implement the solution procedure.

STRUCPROC — STRUCPROC is the controller for the whole program. It consists of a single 'DO loop' that repeats once for each time step. Inside the main loop, it calls the other various procedures that solve the equations for each time step.

A.2 PROBLEM SOLVING ROUTINES USED BY STRUCPROC

AMIDONE — Calls the output routine, RESULTS, after the first (elastic) pass. The name refers not to 'am I done the whole problem', but 'am I done the iterations for one time step'. However, the majority of forseeable uses involve only one iteration per time step.
CALLS: RESULTS, FOSCALC
CALLED BY: STRUCPROC

BMATRIX — Evaluates the strain matrix 'B' at any point within an element.
CALLS: Nothing
CALLED BY: INDATA, KTAN

BCUPDATE — Called only when the input requests boundary condition updates; it handles this entire function.
CALLS: Nothing
CALLED BY: RHSINCREM

CHECK — Acts as a 'filter' for input data, checking the geometry (nodal coordinates and connectivity), boundary conditions, material properties, and it prints a message and stops the program if it finds an internal inconsistency.
CALLS: LISTDAT (if an error is found).
CALLED BY: INDATA

DATABASE — Opens the binary GNP output file (TAPE22), then calls the correct 'PLOT' output routine to produce each successive output to the file each time RESULTS is called to output the same values to paper.
CALLS: PLOT8
CALLED BY: RESULTS

DBMTX — Multiplies the elasticity matrix "D" by the strain displacement matrix "B".
CALLS: Nothing
CALLED BY: KTAN

DTANG — This routine is used by all the constitutive creep laws to create the tangent form of the material property matrix.
CALLS: Nothing
CALLED BY: EXPTIME, NORTON, POWR

DTCRIT — This finds the critical time step which is as large as possible while allowing an acceptably accurate strain rate.
CALLS: POWR OR NORTON
CALLED BY: MATSUB

EMAT — Finds the elasticity matrix for each material type once, at the start of the program.
CALLS: INVERSE
CALLED BY: GEOROC

EXPTIME — Computes inelastic strain if the exponential time law is used.
CALLS: DTANG, HY[illegible]
CALLED BY: MATSU[illegible]

EXTEND — Calculates the centre "fictitious node" position of an element, and writes it to GNP file opened by DATABASE.
CALLS: Nothing
CALLED BY: PLOT8

FOSADD — This routine is called at the end of each time step to sum up a value proportional to the incremental strain, which is used at the end of the modelling to calculate a strain-related factor of safety. (See FOSCALC below).
CALLS: Nothing
CALLED BY: STRUCPROC

FOSCALC — This routine uses the summed up strain-related values, and the final values of 'sigma x', 'sigma y', 'sigma z' to calculate a 'factor of safety' value at the end of the modelling for each Gauss point.
CALLS: Nothing
CALLED BY: STRUCPROC, AMIDONE, RHSINCREM (any subroutine that can halt the model must call FOSCALC at that point).

FORCE — Calculates the element stresses at the end of each time step.
CALLS: MATSUB, SHAPEFN, JACOB, BMATRIX
CALLED BY: RHSINCREM, RHSUPDATE

FRONT — Frontal solver (documented in Hinton and Owen, 1980), it forms the global stiffness matrix, and load vector, and solves this equation system by Gaussian elimination.
CALLS: Nothing
CALLED BY: SOLVER

GAP — Decides whether element material types should be changed based on volumetric closure and if so, changes them
CALLS: VOLUME, INTGRPT
CALLED BY: INDATA, RHSINCREM

HEAD — Assembles the title-time-date string that is printed at the top of each output page.
CALLS: DATE, TIME (These are system dependent functions)
CALLED BY: INDATA

HISTF — A function which calculates the history function used in the exponential time constitutive law.
CALLS: Nothing
CALLED BY: EXPTIME

INDATA — This controls all input to the program except the first card. It performs some error checking, echoes the input and generates missing nodes and elements.
CALLS: TIME, DATE (These are system functions)
MIDSIDE, INTGRPT, CHECK, SHAPEFN, JACOB, BMATRIX, NEGATE, VOLUME, GAP, HEAD
CALLED BY: GEOROC

INSHAPE — This computes the interpolation polynominals used to compute values at the nodal points from those at Gauss points.
CALLS: INVERSE, SHAPEFN
CALLED BY: NEGATE

INTGRPT — Sets up Gauss points and weighting factors.
CALLS: Nothing
CALLED BY: GAP, INDATA

INVERSE — Inverts a matrix and calculates its determinant.
CALLS: Nothing
CALLED BY: GAP, INDATA

JACOB — Calculates the Jacobian matrix and shape function derivatives.
CALLS: Nothing
CALLED BY: FORCE, INDATA, KTAN

KILL — Used when excavating to change the material type of the excavated elements to the properties of 'air', i.e. no strength, and then it zeros the appropriate forces and displacements.
CALLS: Nothing
CALLED BY: RHSINCREM

KTAN — This does the formulation of the tangential stiffness matrix for each element.
CALLS: SHAPEFN, BMATRIX, JACOB, DBMTX
CALLED BY: STRUCPROC

LEFTJ — This left justifies any string, i.e. removes all leading blanks.
CALLS: Nothing
CALLED BY: HEAD

LISTDAT — Reads in and echos all remaining input data after a halt due to errors encountered during input.
CALLS: Nothing
CALLED BY: CHECK, SCRUT

LOADINC — This increments the loads for the incremental material models.
CALLS: Nothing
CALLED BY: RHSINCREM

MATSUB — Controls calls to the material subroutines depending upon the behaviour specified (elastic creep).
CALLS: POWR, EXPTIME, NORTON, DTCRIT
CALLED BY: FORCE

MIDSIDE — This interpolates the positions for the middle of the sides nodes.
CALLS: Nothing
CALLED BY: INDATA

NEGATE — Computes the nodal forces required to set the stresses on a surface to prescribed values, including zero. Used for excavations, with or without internal pressure.
CALLS: INSHAPE, SHAPEFN, STNODS
CALLED BY: INDATA, RHSINCREM

NORTON — Computes the inelastic creep strain based upon the Norton power law.
CALLS: INVERSE, DTANG
CALLED BY: DTCRIT, MATSUB

NULL — Initializes load, stress, strain, strain, displacement and some other arrays to zero, and some 'counter-variables' to one or zero, as needed.
CALLS: Nothing
CALLED BY: GEOROC

PACK — This subroutine is rarely used. It "packs" the stress array into less file space, which the GNP package can still read, if so instructed as an option which is worthwhile only for huge files.
CALLS: Nothing
CALLED BY: PLOT8

PARAMETERS — This subroutine reads the first input card which contains array sizing values.
CALLS: Nothing
CALLED BY: GEOROC

PLOT8 — If the 'database' is requested on the first card, this routine is called each time results ar printed to output the same values to the binary database file, based upon eight-noded elements.
CALLS: EXTEND, PACK, RZ
CALLED BY: DATABASE

POWR — Computes the inelastic creep strain using the transient power law.
CALLS: DTANG, INVERSE
CALLED BY: DTCRIT, MATSUB

RESULTS — This routine controls virtually all output except the input echoing done by INDATA. It prints the tables of displacements and stresses at each output time.
CALLS: DATABASE
CALLED BY: AMIDONE, STRUCPROC

RHSINCREM — This prepares the 'Right Hand Side' of the main equation for processing by FORCE, including checking if it is time for excavation or backfill, and implementing them, before calling FORCE.
CALLS: BCUPDATE, FORCE, FOSCALC, GAP, KILL, LOADINC, NEGATE, TIMECHK
CALLED BY: STRUCPROC

RHSUPDATE — Calculates the stresses, strains, and strain rates extant at the end of a time step, and the loading to be applied on the next step.
CALLS: BMATRIX, FORCE, JACOB, SHAPEFN, STRESS, STRNDIS
CALLED BY: STRUCPROC

RZ — This is used to write out nodal coordinates to the GNP database.
CALLS: Nothing
CALLED BY: PLOT8

SHAPEFN — This is a general purpose function that calculates the shape

functions and their derivatives at a point within an element.
CALLS: Nothing
CALLED BY: FORCE, INDATA, INSHAPE, KTAN, NEGATE, RHSUPDATE, VOLUME.

SIZE — This subroutine "maps" the "dynamically dimensioned" arrays into successive ranges within the master array 'ARRAY'.
CALLS: Nothing
CALLED BY: GEOROC

SOLTYPE — Sets a flag which indicates which solution method will be used for inelastic creep strain.
CALLS: Nothing
CALLED BY: STRUCPROC

SOLVER — Controls the call to the frontal solver.
CALLS: FRONT
CALLED BY: STRUCPROC

STNODS — This calculates the 'xx', 'yy', and 'xy' stresses at a nodal point on a particular surface.
CALLS: Nothing
CALLED BY: NEGATE

STRESS — This computes stresses on an integration point basis.
CALLS: Nothing
CALLED BY: RHSUPDATE

STRNDIS — This computes total and incremental strains.
CALLS: Nothing
CALLED BY: RHSUPDATE

TIMECHK — This subroutine checks whether the next time step will take the simulation past the time of a specified event, such as excavation, backfill, or a printout of results. If so, it

readjusts that time step to just take the total time to the specified value, and sets a flag to make the specified event(s) occur.

CALLS: Nothing

CALLED BY: RHSINCREM

VOLUME — This computes the current volume closure of an element, and is used to decide whether it has reached the time to backfill the element.

CALLS: JACOB, SHAPEFN

CALLED BY: RHSINCREM

APPENDIX B

USER'S MANUAL FOR PROGRAM GEOMESH

1 INTRODUCTION

This document briefly describes program GEOMESH which was designed for geomechanics applications using the finite element method (FEM). GEOMESH is written in FORTRAN 77 and employs the concept of mapping via isoparametric shape functions (Zienkiewicz, 1977, p. 204). The program is quite flexible and allows a variety of complex meshes to be generated with relative ease. It allows the density of elements to be graded from very dense in the regions of concern to very sparse at the mesh perimeters, thus, minimizing the comptutational time required for the FEM solution. GEOMESH also allows boundary conditions to be entered in a straight forward manner. User comments are invited on both this manual and the program itself so that shortcomings and errors can be identified and remedied.

2. MESH GENERATION BY MAPPING (OVERVIEW)

The generation of two-dimensional meshes composed of 4-, 8- or 9-node element topologies is considered. By generation, it is implied that nodal point coordinates and element topologies (connection arrays), as well as material identification, boundary conditions and field variables are generated.

An example mesh is shown in Figure 1. It is seen how multiple maps may be used in combination to create complex meshes with ease. To effect mapping, basically two sets of data are necessary:

(1) Control Points;

(2) Edge Divisions.

Control points, which may be from 4 to 9 in number, serve to bound the region to be mapped. The first four, describing a quadrilateral, are the minimum required. The next four (5-8) are midside control points and are used if the corresponding edge is not a straight line. They may also serve to weight the mesh division towards one corner or the other. For curved edges, this is not a satisfactory method for weighting so a weighting parameter is available. The ninth control point serves to give the interior of the map a particular character. In general, it will not be used. The elements generated will follow the same convention (nodal point numbering) as the maps. The subroutine "UFORMT", which is used to output the final mesh data, may be easily altered by the user to whatever numbering format is desired.

Three alternatives are available for describing the edge divisions. A restriction is that opposite edges must have the same number of divisions, i.e., mesh transition is a feature not accommodated as of yet. The first technique merely states the number of element divisions along the edge and, if required, a weighting may also be included. The second technique simply inputs coordinates of the element divisions. Finally, the last technique uses the divisions of a previous map edge. Once the control points and edge divisions are identified, the element topologies and coordinates may be generated.

In addition to coordinates, nodal point temperatures may be mapped. The values coresponding to the control points, are input. Boundary conditions along map edges as well as element material numbers per map are easily prescribed. A final instruction, which equivalences the current map to a previous map is described on the input card QMAP.

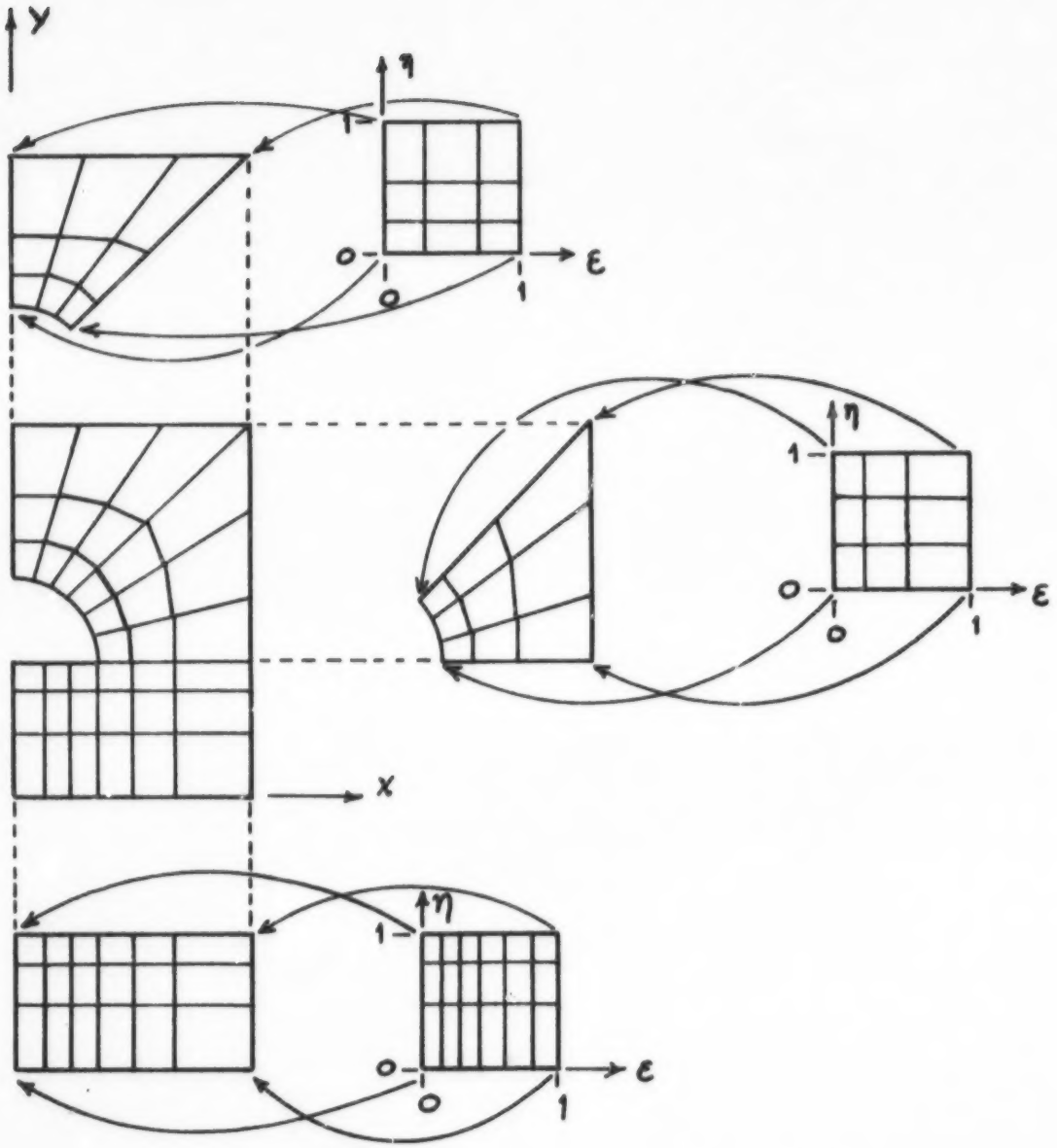


Figure 1. Example of Mesh Generation by 3 Maps from Unit Squares in the ε-η Coordinate System to the x-y Coordinate System.

Updating of nodal coordinates and element topologies is also provided for. This feature, in general, is not particularly attractive unless immediate raster graphics is available, displaying nodal point and element numbering on the mesh.

3. PROGRAM FILES AND EXECUTION

In the main program segment (MAIN), the input/output files are identified and opened. The current conventions of these files are as follows:

LIN = input file no. (called INPUT if batch execution)
LOUT = output file no. (called OUTPUT)
LKEYBD = keyboard file no. (file 0 on Honeywell System).

Interactive mode (input is prompted for at the terminal) is enabled when LIN = LKEYBD = 0; otherwise, LIN > 0. The file 'OUTPUT' contains diagnostics if batch execution. It also contains assorted information such as number of nodal points, number of elements, number of mechanically restrained nodal points and solution resequencing information with resulting nodal frontwidth and maximum bandwidth. The resulting mesh is recorded in two separate files:

NFILEF = mesh for field analysis (SPECTROM-41)
NFILEM = mesh for mechanical analysis (SPECTROM-32)

The solution resequencing strategy follows that outlined by Hoit and Wilson (1983). The algorithm essentially orders the elements based on a minimum front criteria. As a by-product, the nodal point numbers as they come off the front are taken as the new nodal point order. The algorithm is very fast and produces a very good element order, though in general not quite optimum, for a front solution. The related publication suggests that the new nodal point numbering also produces a good order for variable band solution. The maximum band, however, is far from optimum. For such use, the program should avoid the call to subroutine REORDR, and the band should be considered during mapping. The appendix shows the numbering convention for nodal points.

Graphic capability is a necessity for displaying generated meshes. Such plots should be inspected carefully for errors such as superimposed nodal point numbers, which indicates errors in equivalencing. A temporary plotting subroutine (PICTUR) has been included in the program to aid in debugging it. It has only limited 'windowing' capability but was not further developed since the more capable NCAR graphics system will inevitably replace it.

4. INPUT CONVENTIONS

All the data during mesh generation is entered via a free field card reader. The particular version is a modified version of that reader found in the computer program SPECTOM-31 by Dr. S.W. Key. Each data set, which may be more than one line long, is to be preceded by a valid keyword (END, HELP, UPDATE or QMAP). Succeeding data is order independent with the restriction that numeric data is associated with the first alphanumeric descriptor. The following conventions are to be adhered to:

(a) 80 characters maximum per line (not necessarily per data set);
(b) ampersands (&) anywhere in a data line indicate that the following line is a continuation of the current line;
(c) asterisks (*) first on data line indicate coments which are ignored;
(d) blank lines are ignored;
(e) keywords, descriptors and data may be separated by commas, equal signs and/or spaces;
(f) consecutive commas imply zeroes between.

4.1 General Input Keywords

With the exception of HELP, the following keywords require descriptors on a common input card:

HELP - provides a brief description of keyword commands.
UPDATE - provides means for inputting or updating nadal points and elements.
QMAP - effects mesh generation by mapping.
END - terminates mesh generation; renumbers nodai points and elements, deleting unused numbers; renumbers elements and nodal points based on minimum front criteria (should also provide a satisfactory numbering for variable band solution); and outputs mesh data as per user subroutine UFORMT.

As a final step (after END), a '0' input is required for skipping of plotting and a '1' for plotting. Regardless of the input mode (from keyboard or from an input file), graphing is assumed interactive.

4.2 Input Card for Regular Mapping

QMAP, p+1[a], PTi,0[b],0[c], NDIVI, 1[d], NDIV2, 1[e], WGT i, .5[f], &
BCF i,0[g],0[h], BCMi,0[j],0[k],0[l],0[m], EQi, &
MAXNOD,8[n], MATF,1[o], MATM,1[p]

LEGEND:
i = point no. or edge no. - must trail corresponding word without a space.
underscored items require numeric input unless the default values shown above are satisfactory.
p = highest previous input as the default; $p_0 = 0$.

QMAP: keyword effecting mapping operations; [a] map no.
PTi: control point i with [b]x and [c]y coordinates.
NDIV1: [d] element divisions in ξ-direction.
NDIV2: [e] element divisions in η-direction.
WGTi: [f]weighting for uniformly varying element divisions; equals fraction of distance along edge i at which middle of element divisions is located [h](.01 < WGT < .99).
BCFi: field boundary condition [g]code and [h]value.
BCMi: mechanical boundary condition [j],[k]codes and [l],[m]values for x and y components, respectively.
EQi: equivalencing of edge i to previously generated edges.
MAXNOD: [n]nodal points per element (4, 8 or 9).
MATF, MATM: [o]field and [p]mechanical element material numbers.
EDGEi: [q]x and [r]y coordinates (excluding corners) of edge i.
USEi: use edge s of previous map t for current edge i.

Alternatively, in lieu of NDIV1, and NDIV2 and WGTi, a given edge may be described by input coordinates (which must be consistent with mapping from the control points) or by using the edge of a previous map.

EDGEi, q, r, q, r, q, r,

or USEi, s, t

Additional notes and restrictions for regular maps:

(a) Opposite edges must have equal element divisions.

(b) Input coordinates for edges must correspond to control points, i.e., mapping is based on control points with edges serving as divisions only. Practically, the coordinate input feature (EDGEi) is restricted to straight edges.

(c) When using WGTi, if the opposite edge is not given a weight, 1-WGTi will be assumed for that edge, i.e., consistent weighting, will be realized in the corresponding direction.

The following descriptors may also be included on the QMAP card.

TEMP, 0^u, 0^u, 0^u, 0^u, 0^u, 0^u, 0^u, 0^u, 0^u, NODE, $p+1^v$, ELEMENT, $p+1^w$

TEMP = temperatures^u at control points
NODE = beginning node no.^{v} (useful for updating)
ELEMENT = beginning element no.^{w} (useful for updating)

Node and element default beginning numbers are one greater than highest previous used numbers.

4.3 Card for Updating/Input of Nodes or Elements

UPDATE, NODE, 1^a, e^b, e^c, BCF, e^d, e^e, BCM, e^f, e^g, e^h, & e^i &
ELEMENT, 1^j, e^k, e^k, e^k, e^k, (e^k, e^k, e^k, e^k, e^k), MATF, e^l, MATM, e^m

The sequences NODE... and/or ELEMENT... may be repeated a number of times on a given UPDATE card.

Legend: e = existing value is taken as default.

UPDATE: keyword permitting updating/input of nodes or elements.

NODE: node no. a is given coordinates x^c and y^c, field boundary conditon code d with value e, mechanical boundary condition codes f and g with corresponding values h and i.

ELEMENT: element no. j is given nodal points (4-9) k, field material no. l and mechanical material no. m.

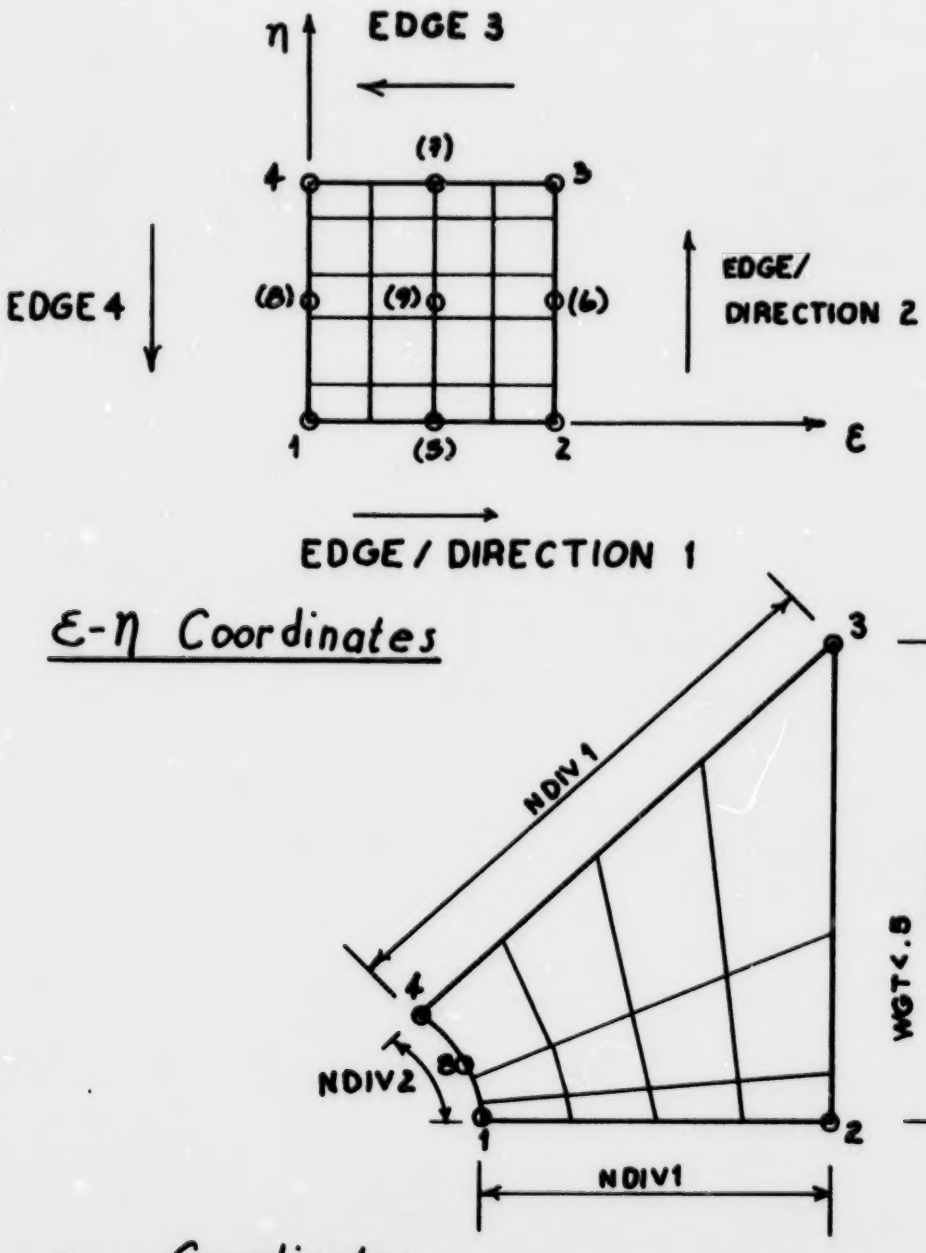
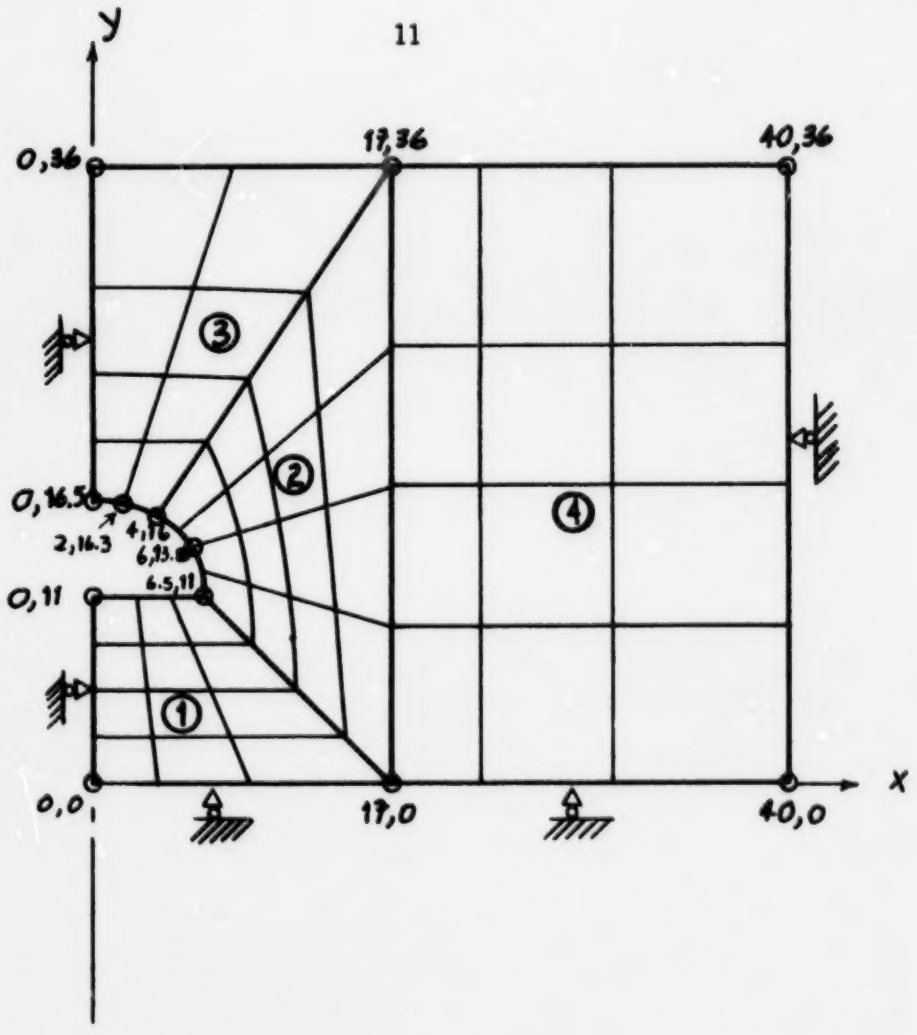


Figure 2. Mapping Conventions.




```
QMAP,1,PT2,17,0,PT3,6.5,11,PT4,0,11,NDIV1,3,NDIV2,4,WGT2,.7, &
     BCM1,0,1,BCM4,1
QMAP,2,PT1,6.5,11,PT2,17,0,PT3,17,36,PT4,4,16,PT8,6,13.8,NDIV1,4, &
     NDIV2,4,WGT1,.3,EQ1
QMAP,3,PT1,0,16.5,PT2,4,16,PT3,17,36,PT4,0,36,PT5,2,16.3, &
     NDIV1,2,NDIV2,4,WGT2,.3,EQ2,BCM4,1
QMAP,4,PT1,17,0,PT2,40,0,PT3,40,36,PT4,17,36,NDIV1,3,NDIV2,4, &
     WGT1,.3,EQ4,BCM1,0,1,BCM2,1
END
1    (Flag Set for Plotting; Otherwise=0)
```

Figure 3. Example Problem 1 - Rough Sketch and Input data for Mesh Generation (Mechanical Analysis); (Isolated Excavation below Ground Level).

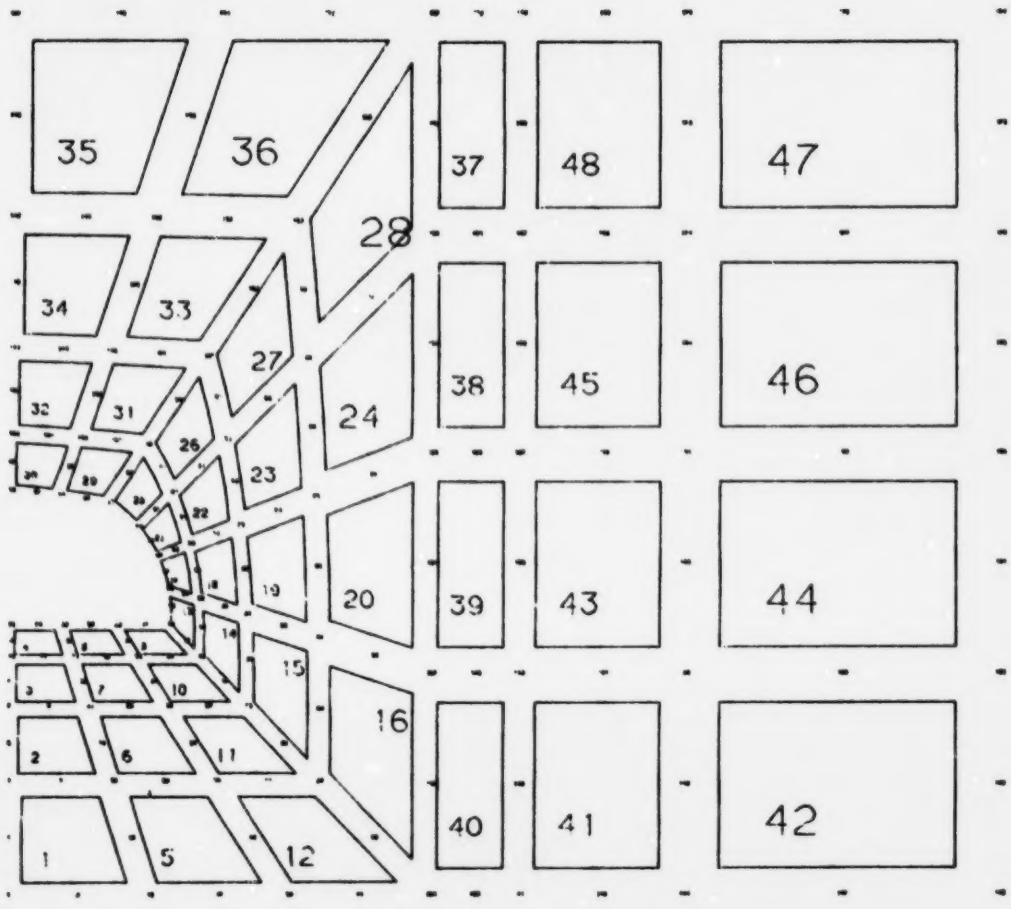


Figure 4. Mesh for Problem 1.

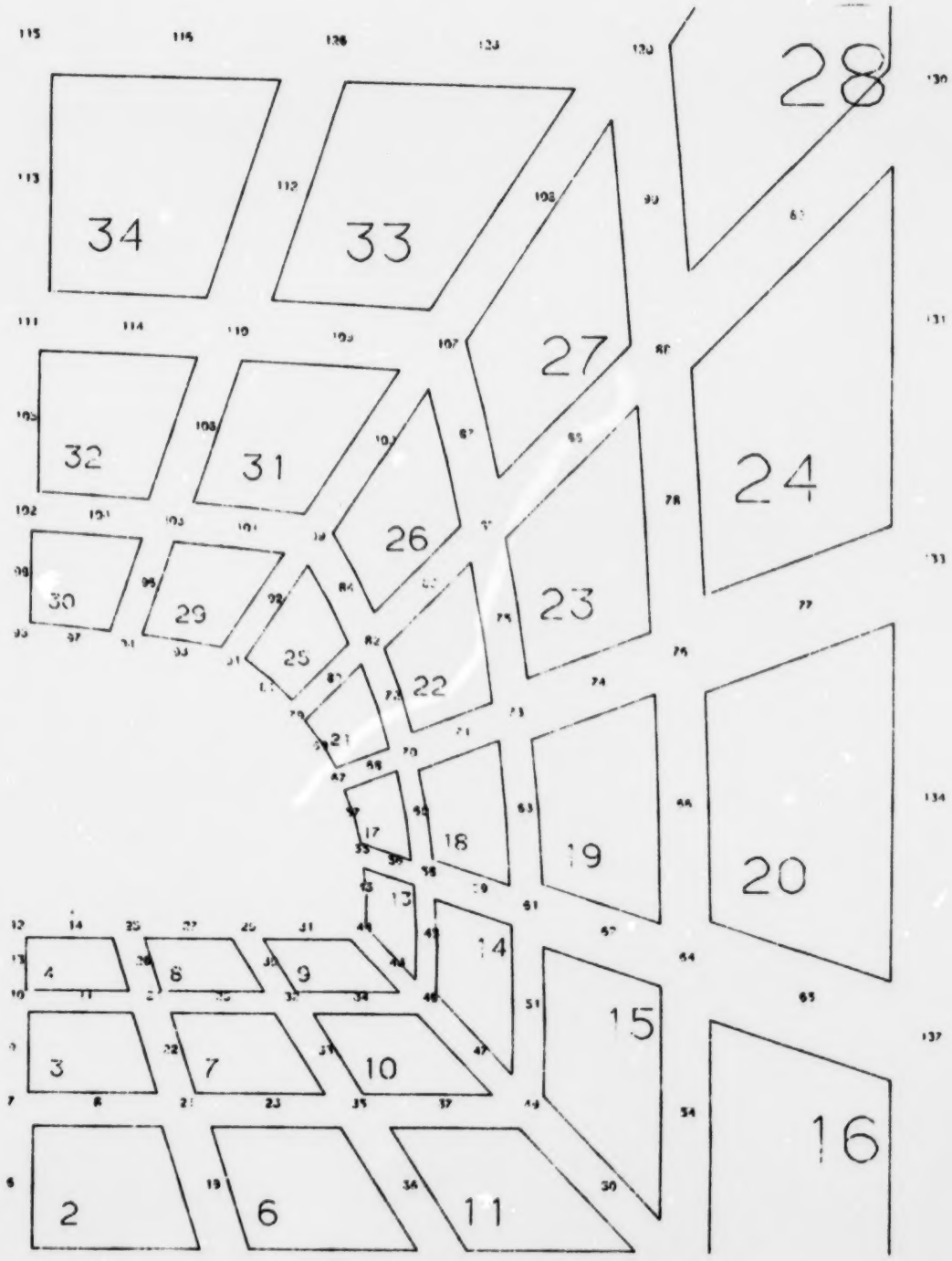


Figure 5. A Detailed View of the Mesh for Problem 1.

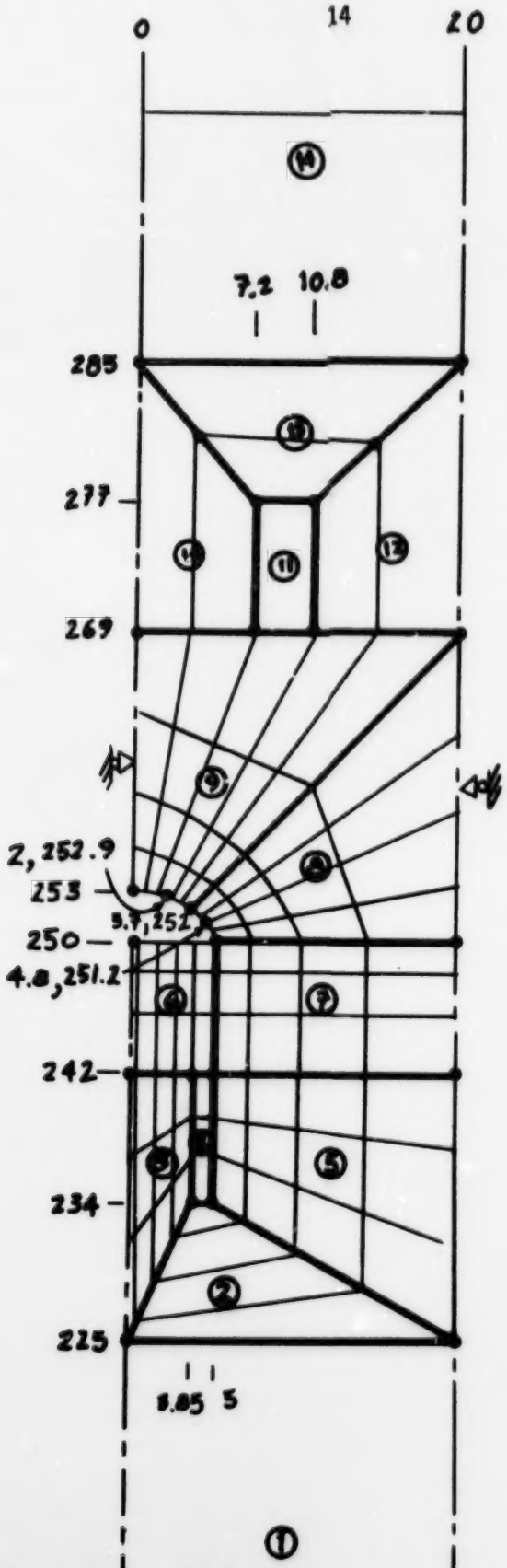


Figure 6. Sketch of Example Problem 2 Showing Blocks (Maps) Used in Input Data.

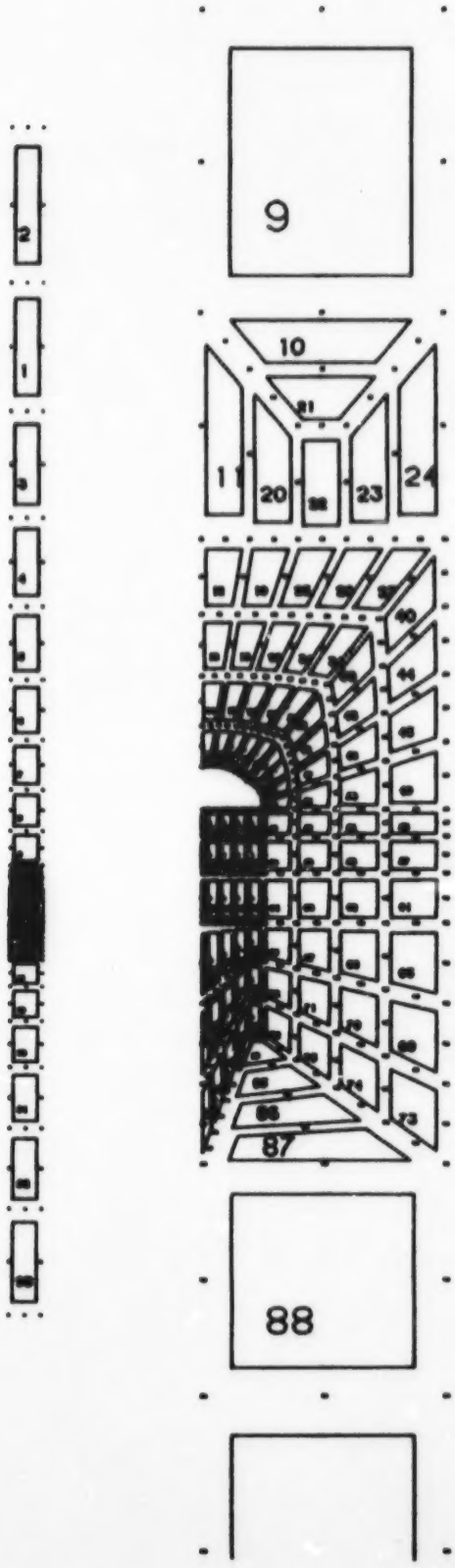


Figure 7. Mesh for Example Problem 2 (See Table 1 for Input Data).

Table 1. Input Data Required for Example Problem 2

```
QMAP,1,PT2,20,PT3,20,225,PT4,0,225,NDIV2,6,WGT2,.78
     BCM1,0,1,BCM2,1,BCM4,1,BCF2,1,BCF4,1

QMAP,2,PT1,0,225,PT2,20,225,PT3,5,234,PT4,3.85,234,NDIV2,48
     WGT2,.6,WGT4,.5,EQ1

QMAP,3,PT1,,225,PT2,3.85,234,PT3,3.85,242,PT4,0,242,NDIV1,4,8
     NDIV2,3,EDGE3,2.7,242,1.55,242,.4,242,EQ1,BCM4,1,BCF4,1

QMAP,4,PT1,3.85,234,PT2,5,234,PT3,5,242,PT4,3.85,242,NDIV2,38
     EQ1,EQ4

QMAP,5,PT1,5,234,PT2,20,225,PT3,20,242,PT4,5,242,NDIV1,48
     NDIV2,3,WGT1,.4,EQ1,EQ4,BCM2,1,BCF2,1

QMAP,6,PT1,0,242,PT2,5,242,PT3,5,250,PT4,0,250,NDIV1,5,8
     NDIV2,3,WGT2,.6,EDGE1,.4,242,1.55,242,2.7,242,3.85,2428
     EDGE3,3.85,250,2.7,250,1.55,250,.4,250,EQ1,BCM4,1,BCF4,1

QMAP,7,PT1,5,242,PT2,20,242,PT3,20,250,PT4,5,250,NDIV1,4,NDIV2,38
     WGT1,.4,WGT3,.6,WGT2,.6,EQ1,EQ4,BCM2,1,BCF2,1

QMAP,8,PT1,5,250,PT2,20,250,PT3,20,269,PT4,3.7,252.2,PT8,4.8,251.28
     NDIV1,4,NDIV2,4,WGT1,.4,EQ1,BCM2,1,BCF2,1

QMAP,9,PT1,0,253,PT2,3.7,252.2,PT3,20,269,PT4,0,269,PT5,2,252.98
     NDIV1,5,NDIV2,4,WGT2,.4,EQ2,BCM4,1,BCF4,1

QMAP,10,PT1,0,269,PT2,8,269,PT3,8,277,PT4,0,285,NDIV1,2,EQ1,BCM4,18
     BCF4,1

QMAP,11,PT1,8,269,PT2,12,269,PT3,12,277,PT4,8,277,EQ1,EQ4

QMAP,12,PT1,12,269,PT2,20,269,PT3,20,285,PT4,12,277,NDIV1,2,EQ1,EQ48
     BCM4,1,BCF4,1

QMAP,13,PT1,8,277,PT2,12,277,PT3,20,285,PT4,0,285,NDIV2,2,EQ1,EQ2,EQ4

QMAP,14,PT1,0,285,PT2,20,285,PT3,20,750,PT4,,750,NDIV2,9,EQ1,BCM2,18
     BCM4,1,BCF2,1,BCF4,1,WGT2,.3,

END
1 (flag set for plotting; otherwise = 0)
```

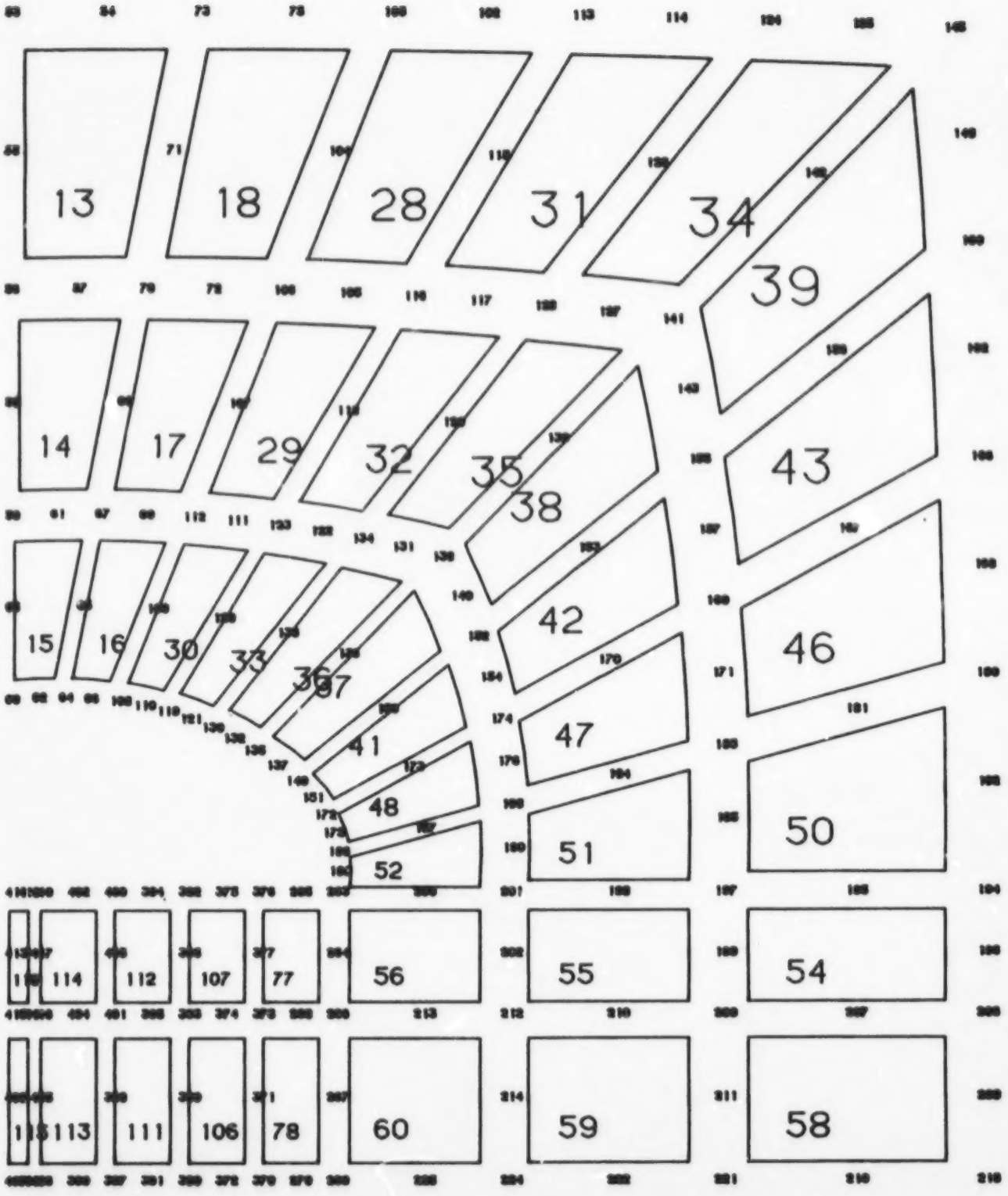


Figure 8. Detail of Mesh Created for Example Problem 2 (See Table 1 for Input Data).

APPENDIX A

A.1 Use of Weighting Parameter

Figures A-1 through A-3 demonstrate different aspects of mesh generation. Figure A-1 is a uniform mesh of 9-node elements. The descriptor, MAXNOD, on the QMAP card selects either 4-, 8- or 9- node elements. If omitted, the elements will assume the default, 8 nodes per element. Figure A-2 demonstrates weighting of the element divisons in a given direction. The WGT_ descriptor effects this feature and may be independently specified for each edge. Since opposite edges generally will be weighted similarly, only one edge need be input. If the opposite edge is not weighted, a weighting of 1-WGT_ will be assumed. The reason for 1-WGT_ instead of WGT_ is that the directions, by convention, are opposite. To override this default, both edges are simply weighted. The particular value of WGT (.01 < WGT < .99) correspoonds to the normalized fraction of the distance along the edge at which the middle of the middle element (if odd number of elements) or middle nodal point (if even number of elements) is located. The default for WGT is 0.5.

Figure A-3 demonstrates the effect that input of the ninth node can have. Since it was not placed midway between control points 6 and 8, a distorted mesh, as shown, results.

Finally, common to Figures A-1 through A-3 is that they were generated by a single map with only corner control points 1-4 and midside control points 6 and 8 used to delimit the map.

A.2 Techniques for Mesh Gradation

Figure A-4 shows how a mesh may be graded from very refined in regions of concern to very course near the mesh perimeter. This feature may be accomplished by using appropriate blocks (maps) as shown and can result in considerable saving in computation time.

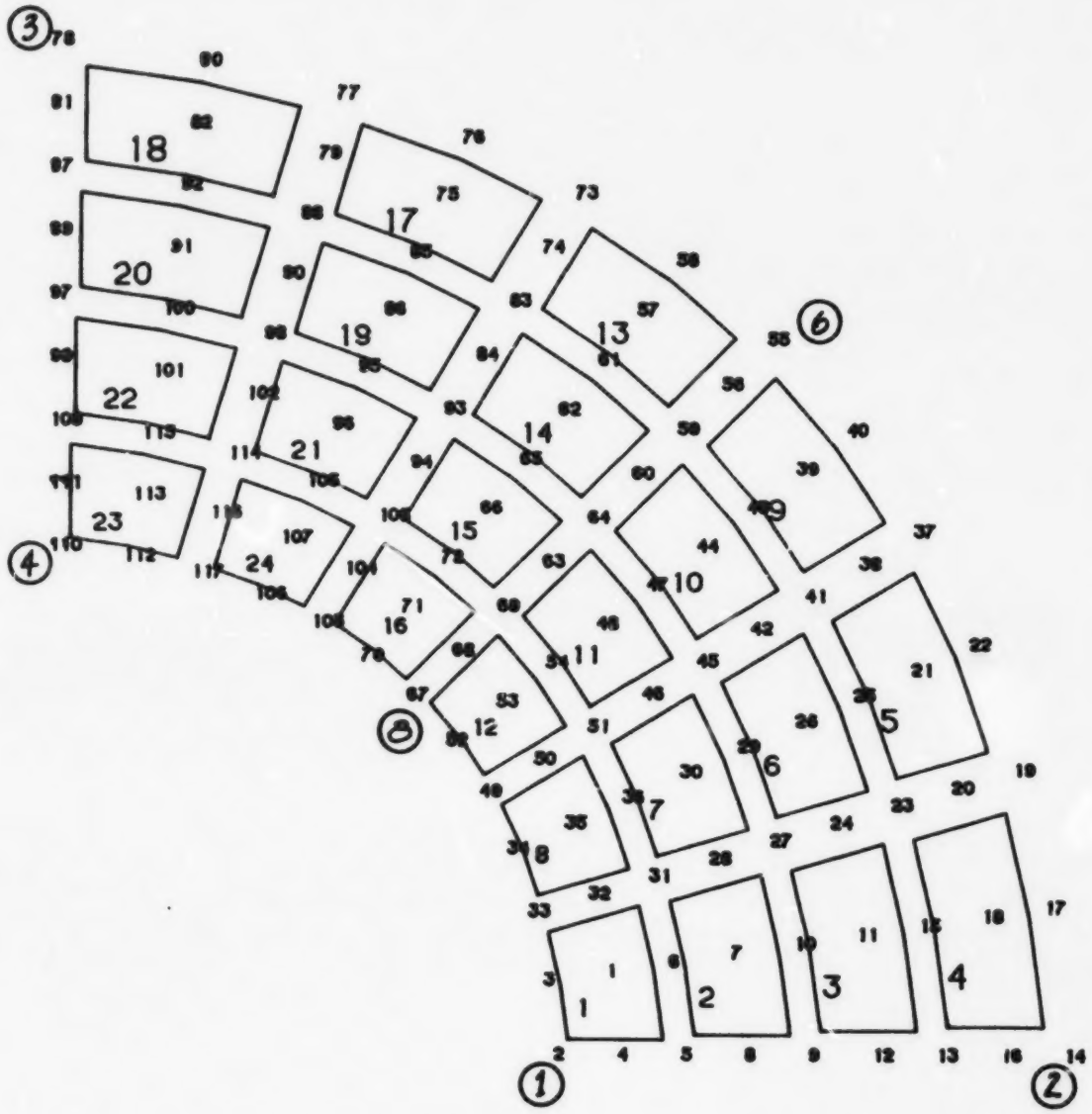


Figure A-1. A Uniform Mesh Composed of 9-Noded Elements.

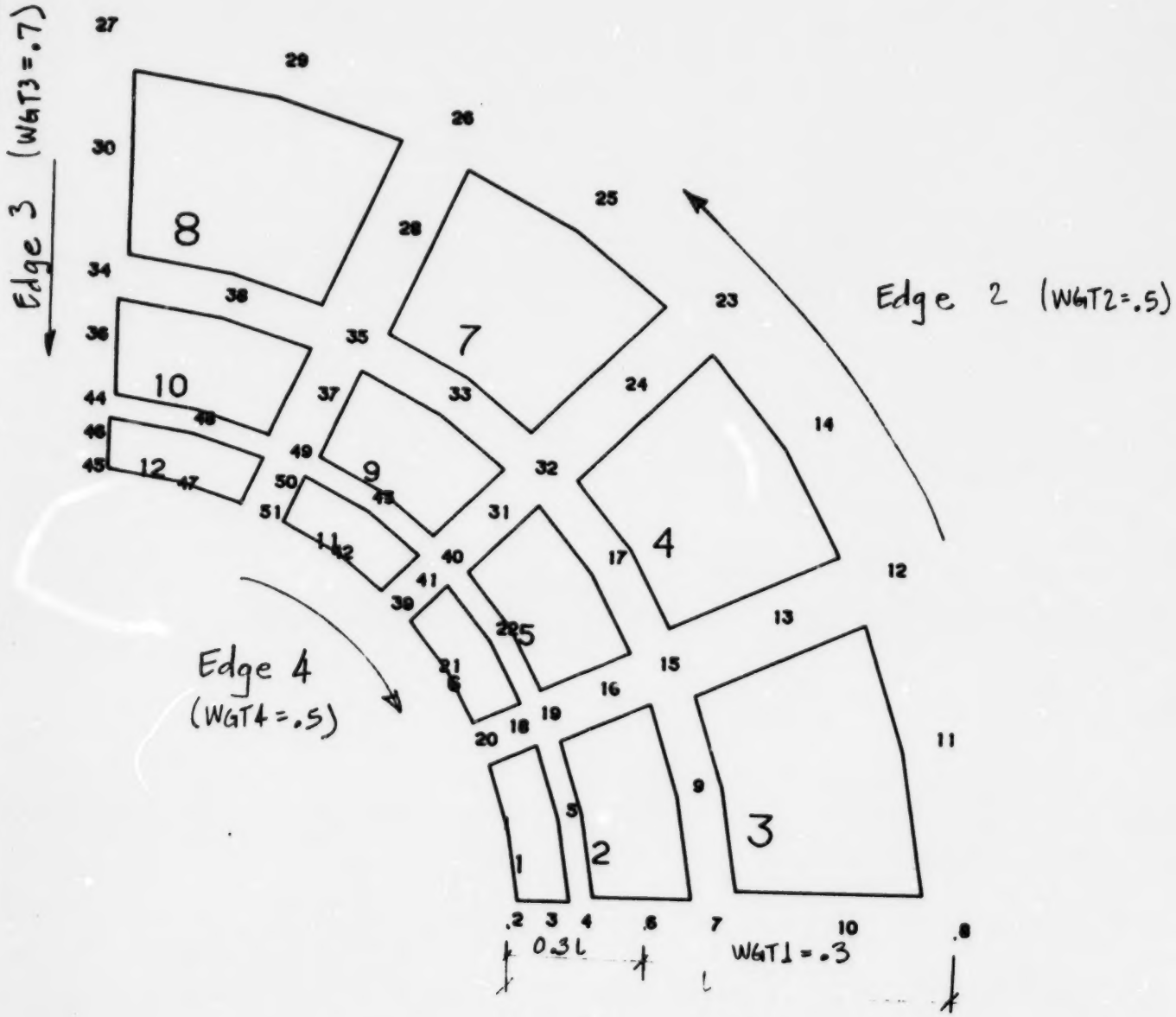


Figure A-2. A Mesh Composed of 8-Noded Elements with Weighting of Element Divisions Using WGT.

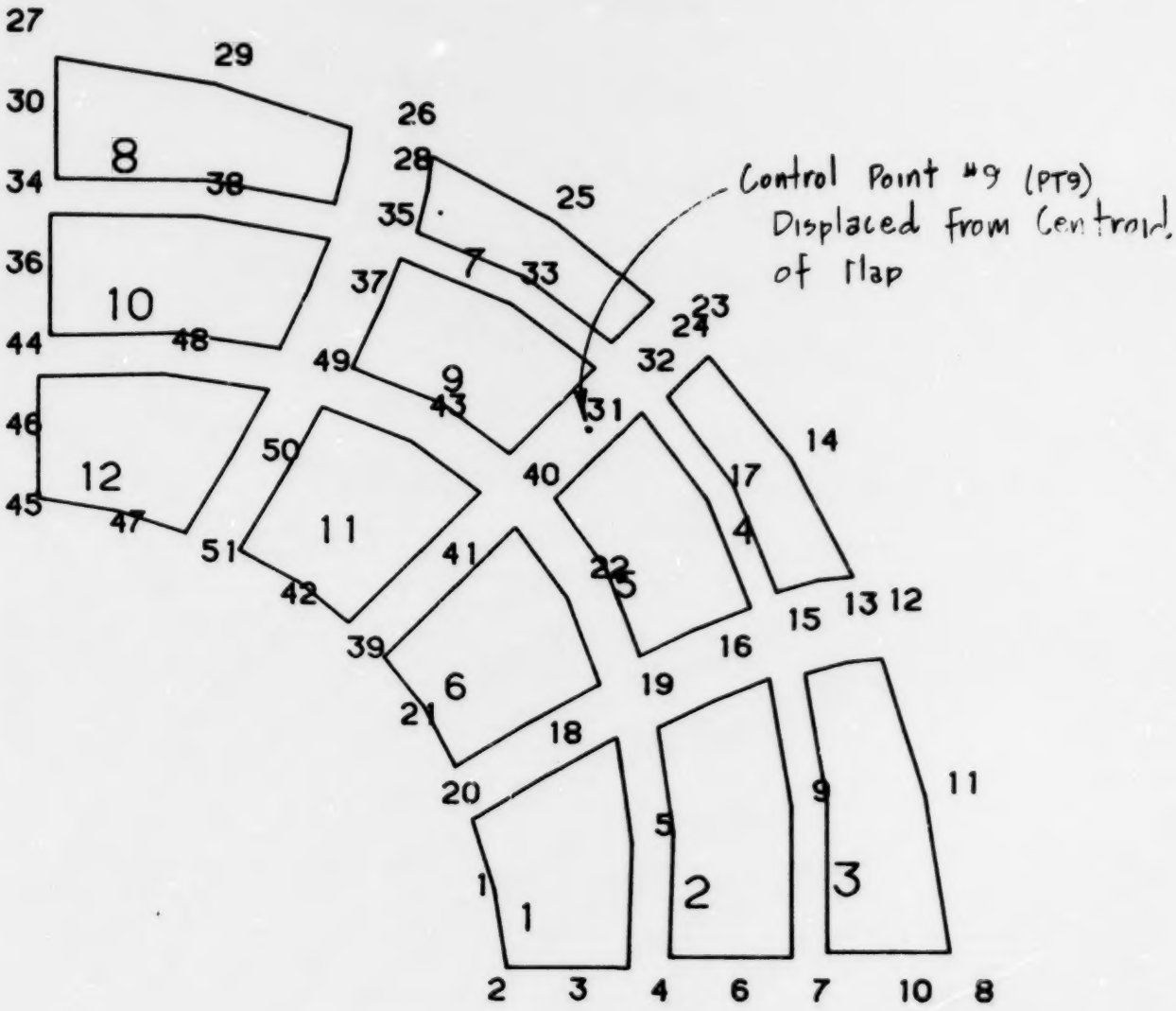


Figure A-3. A Mesh Illustrating Weighting of Element Division Using Control Point 9.

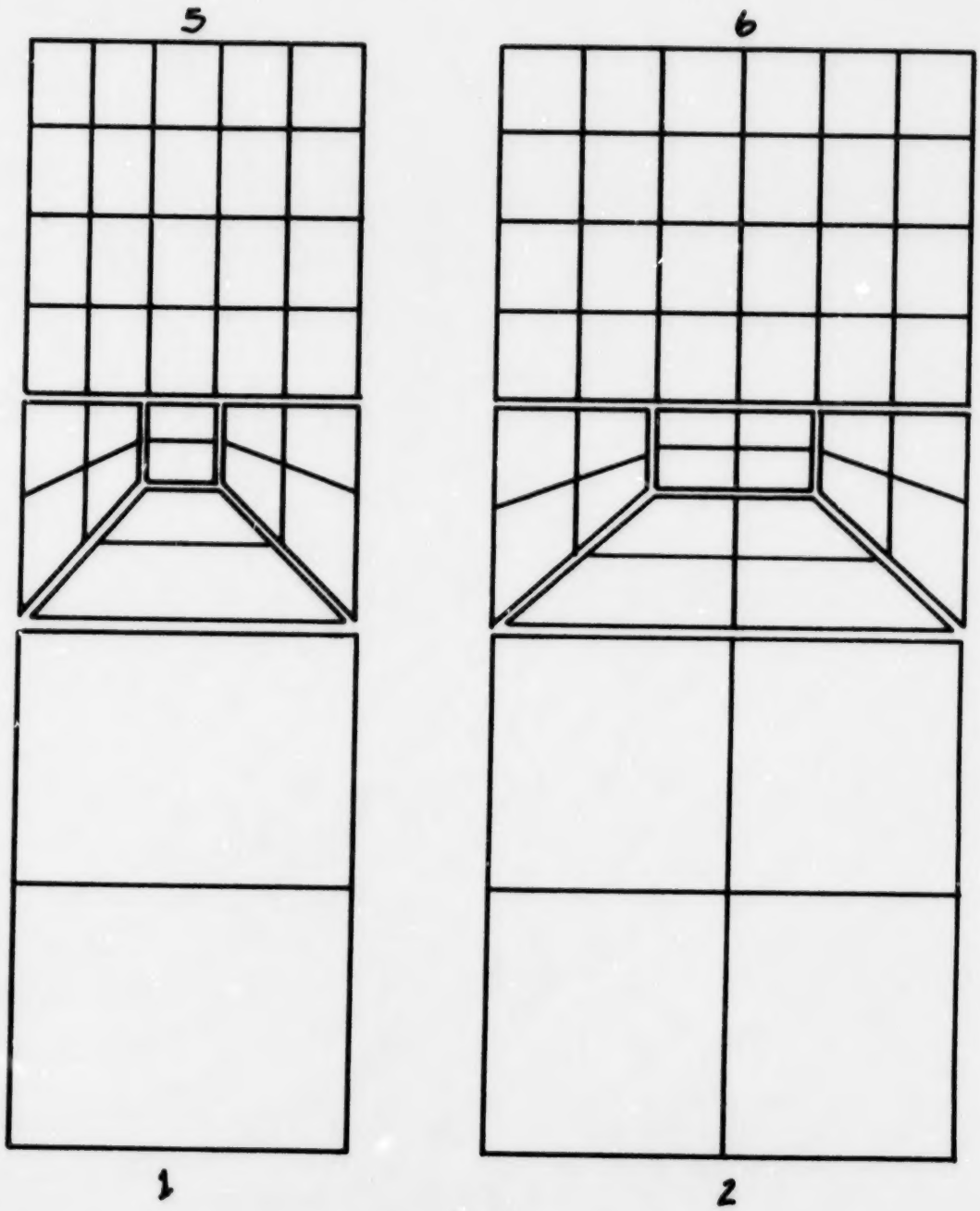


Figure A-4. Illustration of how Mesh Graduation may be accomplished.

A strong motivation for developing more flexible mesh generation strategies is the high solution cost resulting from carrying the fine meshing in the region of the excavation, throughout the entire mesh. Example Problem 2 addresses this concern. The statistics for this problem include:

number of elements = 116
number of nodal points = 417
maximum frontwidth = 29 (nodal)
average frontwidth = 16 (nodal)

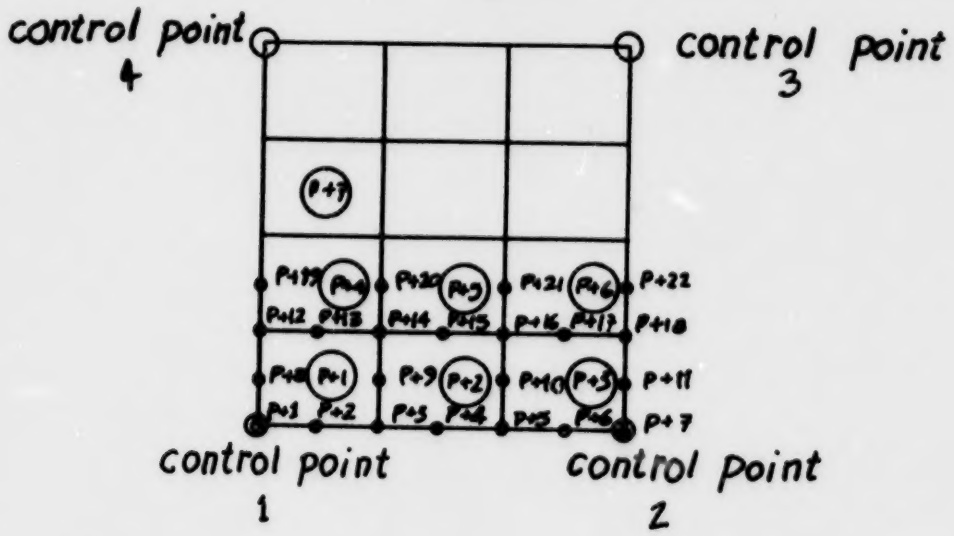
Had the usual mesh discretization at the floor level of the mesh been used throughout the height of the mesh, the following statistics would result:

number of element = 225
number of nodal points = 745 approx.
maximum frontwidth = 24 (nodal)
average frontwidth = 23 (nodal) approx.

Although the solution effort is a function of the root-square of the front-or bandwidth as well as a linear function of the number of equations, a comparison is still afforded since the frontwidths are approximately equal. A ratio between the number of nodal points suggests that a solution at just over one-half the effort of the conventional mesh is afforded by the improved mesh. Of course, the degree to which refinement of the mesh in any given region is adequate is a matter of judgement. With the exception of the surface portion of the mesh in example 2, the refinement is most certainly adequate for the loading considered in typical analyses.

A.3 Numbering Scheme for Nodes and Elements

The numbering scheme for both nodes and elements is shown in the accompanying figure.



p = previous highest element or node numbers.

Note: Boundary nodes, if equivalenced, assume the numbers of the existing nodes. This will not affect the remaining numbering. The result is that some node numbers will be unused during mesh generation. As a final step (after the keyword 'END'), these unused nodes (and elements) are 'squeezed' off the top of the corresponding node or element lists.

A.4 Suggestions for Further Improvement of This Mesh Generator:

(a) Use of blended functions instead of isoparametric functions thereby avoiding isoparametric distortion along curved edges. This would entail replacement of SUBROUTINE SHAPEF.

(b) Addition of another mapping strategy that permits different numbers of element divisions along opposite edges. This would involve an automated transiton strategy within a map and the addition of subroutine similar to SUBROUTINE QMAP.

(c) Mesh optimization within a map. For distorted maps, element shapes become marginal. Via a variational technique applied to the interior

nodes of a map, a system of equations in Δx and Δy for these nodes is generated by minimizing the departure from 90° at all such element intersections.

(d) Immediate raster graphics displaying each stage of the mesh generation, complete with nodal point and element numbering. This would permit convenient updating and input of mesh data. Also boundary conditions and field variables could be edited.